Brno University of Technology
Faculty of Civil Engineering
Institute of Structural Mechanics

2^{nd} PhD Workshop

BRNO-WEIMAR-PRAGUE-WIEN$_{BOKU}$

Book of Abstracts

November 20-22, 2003
Brno, Czech Republic

Brno University of Technology
Faculty of Civil Engineering
Prof. Petr Štěpánek, dean
Institute of Structural Mechanics
Prof. Drahomír Novák

Bauhaus-Universität Weimar
Civil Engineering
Institute of Structural Mechanics
Prof. Christian Bucher

Czech Technical University in Prague
Faculty of Civil Engineering
Department of Structural Mechanics
Prof. Zdeněk Bittnar

Universität für Bodenkultur Wien
Institute for Structural Engineering
Prof. Konrad Bergmeister

LOCAL ORGANIZERS

Prof. Drahomír Novák
Dr. Zbyněk Keršner
Petr Frantík
David Lehký

Anna Kolouchová
Dana Kotoučková

GA ČR 103/03/1350

ISBN 80-214-2506-7

Dedicated to the 70^{th} Birthday of Professor Břetislav Teplý

Table of Content

Approaches to Thermo-mechanical Material Modelling of Wood

Dietmar BOBACZ, *Institute of Structural Engineering, University of Natural Resources and Applied Life Sciences Vienna, Austria*

Wood is among the commonly used materials for structural purposes the prototype of an anisotropic material. However the transferability of the "rhombic crystal structure" on wood is applicable. The three anatomical main directions of the wood (grain, tangential direction and radial direction) can be assigned in good approximation to the three principal axes of a right-angled spatial coordinate system.

According to Eurocode 5 the strength of the material is determined by the 5%-fractile at normal temperatures. If wood is subjected under thermal load (e.g. fire), the 20%-fractile will be applied. The internal forces and stresses are normally calculated linear-elastic. Though in case of fire the stress in the cross-section should be better determined using a non-linear model considering the decrease of strength at sections at higher temperatures and the resulting redistribution of stress for a maximum utilization of the strength of the material. As it can be seen in the stress-strain-diagram in figure 1 wood indicates a plastic behavior at compression and a linear stress-strain law at tension.

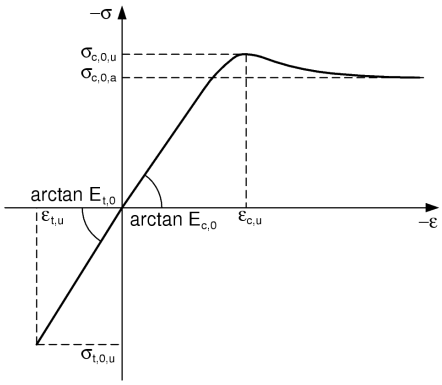


Figure 1: Stress-strain law of wood

To consider the decrease of strength at higher temperatures of wood, the temperature development in the cross-section has to be known. For this purpose a transient temperature field computation with the finite element method is necessary. Yet that is not a trivial task. Beside the mechanical properties, the thermodynamical properties of timber behave non-linear as well. The heat conductivity and the specific thermal capacity are dependant on the temperature, the moisture content, the gross density and the orientation of the grain. Furthermore the pyrolysis and charring of timber has to be implicated which are basically coupled with the gross density and the temperature. Other temperature-dependent characteristics as the creep and the thermal strain of wood can be assumed to be balanced and therefore they need not be taken into account.

The combinations of the non-linear thermal and the non-linear mechanical characteristics, i.e. the stress-strain-laws of compression and tension at normal temperature up to 300C (loss of load-carrying capacity) are the basic input factors for a finite element analysis and represent the fundamental material model of wood. The practical usability of such a model is another problem that should not be underestimated. The implementation in a FE-program asks for a well thought-out algorithm to finally determine the load-carrying capacity of a structural member, e.g. a truss or a beam.

After all data is gathered a last question is still to be answered - the definition of the fire load. It is normally done by using the unit temperature curve or other standardized temperature curves. However, does they really reflect realistic conditions in a specific case of fire? The solution is to combine the finite element analysis with a fire simulation done by a CFD-program (computational fluid dynamics), which provides the actual fire load on the load carrying part. This procedure yields an almost realistic temperature curve for a specific structural part subjected to a fire load. It may be more or less favorable for dimensioning but it is nearer to the real world.

Numerical modelling and sensitivity analysis at the FRP-concrete interface using SARA

Barbara BONFIGLIOLI, *DISTART, University of Bologna, Italy*

Fibre Reinforced Polymer (FRP) plates or sheets externally bonded are one of the most efficient techniques for axial, bending or shear strengthening of structural elements. Due to their useful properties, FRP are finding ever-increasing use in structural engineering. Debonding is one of the most dangerous failure mechanisms. The complexity of the mechanism at FRP-concrete interface influences the local phenomena which have to be correctly predicted in order to obtain a reliable safety evaluation. In literature, some studies concerning these local phenomena can be found. In the last years, different commercial finite element packages are available to model non linear problems, but usually they remain at the deterministic level. Nevertheless, the parameters used are affected by uncertainties so it should be useful to use a software accepting statistical inputs. For this reason the software chosen is FREET-ATENA-SARA. Both an experimental programme and a numerical simulation have been performed. 64 pull-off tests have been carried out with different load angles (from 15 to 90). So a failure domain has been found. From this failure domain, the values of the characteristic parameters (tensile strength, cohesion and friction) have been evaluated. The number of the tests for each angle was enough big to allow us to define the statistic parameters (mean value, standard deviation and coefficient of variation) needed to define a probability density function. Initially, to evaluate the values of the cohesion and the friction, the Coulomb law has been applied to the experimental values, because this law has been adopted for the available interface material model. A difference has been detected between the numerical results and the experimental data. So other failure domains have been taken into account, namely the Mohr-Caquot model and the Leon model. The last one seems to be the most adequate to interpolate all the experimental data[1]. At the end of this step of the research we can conclude that it is possible to define experimentally a failure domain for the FRP-concrete interface and the sensitivity analysis can outline the most significant parameters describing the interface behaviour. Further developments can go in the direction of performing more experimental tests aimed to obtain a data base for the interface parameters.

[1] Bonfiglioli, B.; Strauss, A.(2003).“Sensitivity analysis of FRP-concrete interface and of FRP strain measurement errors using SARA”Festschrift Konrad Bergmeister. 10jähriges Wirken am Institut, 1993 - 2003.Veröffentlichung des Instituts für Konstruktiven Ingenieurbau, pp.12.1 - 12.11,ISBN 3-900962-49-9

The EAS-Method applied on a four-node plane element with internal iteration to model nonlinear material behaviour

Maik BREHM, *Institute of Structural Mechanics, Weimar, Germany*

Iso-parametric finite elements with linear shape functions show in general a to stiff element behavior, called locking. By the investigation of structural parts under bending loading the so-called shear locking appears, because these elements can not reproduce pure bending modes. Therefore many studies dealt with this problem and a number of methods, like the "Assumed Natural Strain" (ANS) method [1] and the "Enhanced Assumed Strain" (EAS) method [2], to avoid locking have been developed.

In this study the EAS-Method is applied to a four-node plane element. There the iso-parametric strain modes will be enhanced through higher-order strain modes. The enhanced strains are modeled as

$$\boldsymbol{\epsilon}_{enh} = \mathbf{M}\boldsymbol{\alpha} \tag{1}$$

where $\boldsymbol{\alpha}$ are the additional enhanced degrees of freedom and $\mathbf{M}$ includes the additional stress-strain-relation, respectively. In this case we are using four EAS-Parameters. After condensing out the enhanced parameters $\boldsymbol{\alpha}$ out of the modified element equational system, we will get the modified element stiffness matrix $\mathbf{K}_{mod}$, which is used instead of the isoparametric stiffness matrix.

The application of the EAS-Method for linear elastic calculations is straightforward. For the research of the subproject A1 of the Collaborative Research Center 524 "Materials and Structures in Revitalization of Buildings", where a coupling between meshless components and finite elements is used [3], nonlinear materials with stochastic distributed properties are the point of interest.

The problem in applying the EAS-Method to physical nonlinear simulations is the nonlinear material matrix $\mathbf{C}$. This matrix depends on the total strain values at the integration points, given by the sum of the iso-parametric strains $\boldsymbol{\epsilon}_{iso}(\mathbf{u})$ and the enhanced strains $\boldsymbol{\epsilon}_{enh}(\boldsymbol{\alpha})$. Furthermore, the enhanced parameters $\boldsymbol{\alpha}$, which are given through the condensation of the stiffness matrix, depend on the material matrix $\mathbf{C}$. This nonlinear problem has to be solved for a given displacement state $\mathbf{u}$. In this study this is done iterative at the element level by using the saved $\boldsymbol{\alpha}$ values from the last step of the global Newton-Raphson iteration as start values. Convergence and the behavior in the global iteration process are the points of interest.

The presented element has been implemented in the SLang Software package, which is available at the Bauhaus-University for research activities.

[1] Simo, J.C. and Hughes, T.J.R. (1986). "On variational foundations of assumed strain methods". Journal of Applied Mechanics, 53:51-54.

[2] Simo, J.C. and Rifai, M.S. (1990). "A class of mixed assumed strain methods and the method of incompatible modes". International Journal of Numerical Methods in Engineering, 29:1595-1638.

[3] Most, Thomas and Bucher, Christian (2003). "Moving Least Squares"-elements for stochastic crack propagation simulations coupled with stochastic finite elements". In A. Der Kiureghian, S. Madanat and J. Pestana, editors. ICASP9 2003, San Francisco, Califonia, July 6-9, 2003. Balkema, Rotterdam, 2003.

Structure-soil interaction – contact stress and settlement

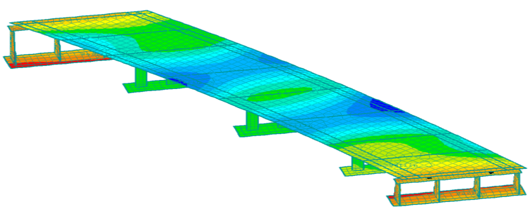
Jiří BUČEK, *FEM CONSULTING, Brno, Czech Republic*

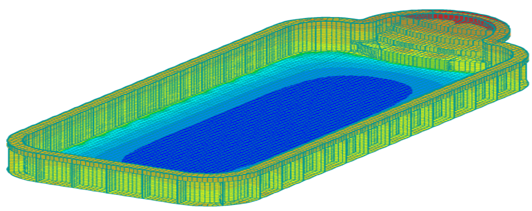
Every building structure must be founded on a subsoil and sometimes this question is told: "is it possible to solve the structure separately and to replace an influence of subsoil by an estimation"? This contribution could be reply to the problem.

For some years I have been working on program SOILIN - Calculation of settlement and parameters of structure-soil interaction [1]. This program can be used either individually to the prognosis of the settlements of a loaded subsoil surface or as a preprocessor to FEM-programs, especially to the NEXIS (ESA PRIMA WIN) program package [1]. In the last case program SOILIN calculates not only settlements but mostly the parameters C of the 2D subsoil model, too.

The program SOILIN uses the Boussinesq solution of the infinite homogeneous and isotropic halfspace for a determination of state of stress in a subsoil and the course σ_z is extra modified with the excavation factor and the factor of underformable layer. The used standards (ČSN, DIN EUROCODE) allow to specify the settlement only from the vertical normal stresses σ_z. Strains ϵ_z are solved generally by a nonlinear physical law $\epsilon_z = f(\sigma_z)$. The function f can be very complicated, e.g. ČSN Standard introduces rigid-elastic stress-strain law using the so-called soil structure strength and DIN Standard assumes an experimentally obtained graphs ϵ_z versus σ_z. These physical subsoil models are included in the program SOILIN, too. In any case the solved problem is strong physical nonlinear. Therefore, no principle of linearity and superposition holds. That is why the solution of structure-soil interaction is necessary to obtain in iteration steps. Firstly the FEM analysis of upper structure with initial C interaction parameters finds out a first approximation of contact stress. These values of contact stress are loading for subsoil and serve as input for the SOILIN. This program solves settlements and corrects values of C parameters. Whole cycle FEM + SOILIN is repeated until the iteration test is fulfilled. In this way correct deformations and internal forces of construction are obtained (see following figures).

On that account it is evident that structures on subsoil is necessary to solve such as the analysis of structure-soil interaction. Loads and own weight can be taken as acting directly on subsoil only then if the foundation and the interacting upper structure are extremely flexible. In any other case the subsoil is overloaded by the contact stress in the foundation-subsoil interface. The contact stress then depends on stiffness of upper structure, loads, geometry of loaded areas, geomechanical characteristics of subsoil, neighbouring building, etc., see [2], [3].





[1] NEXIS program package. User manuals of the program system for the analysis of structures, 2003

[2] Kolář V., Němec I. "Modelling of Soil-Structure Interaction", ELSEVIER, New York, Oxford, London, Amsterdam, Tokyo, 1990

[3] Kolář V., Němec I. "Contact Stress and Settlement in the Structure-Soil Interface". Study of CSAV 17.91, ACADEMIA, Prague, 1991

Structural glazing – Geometrical dimensions and statistic evaluations

Denise EBERSPÄCHER, *Institut of Structural Engineering, University of Natural Resources and Applied Life Sciences, Vienna, Austria*

The objective of the research project "Glass as load carrying element" placed at the Institut of structural engineering is to establish new and innovative fields of application for this building material and break new ground towards intelligent solutions in design-engineering by analysing the findings of recent studies and further ascertaining other essential basic principles. Two main aspects considered during these investigations are the enhancement of structural glazing and the development of cladding systems.

The latest design and development of cladding systems moves more and more towards plane, large surface claddings and as a consequence towards a minimisation of joints. Thus the possibility to compensate certain tolerances and strains in the cladding are restricted. For this reason the construction tolerances and deformations of the structure during its use must be known and accounted for during the design and dimensioning.

In general we may distinguish between two kinds of tolerances: Tolerances as results of the structural works and tolerances in consequence of the production process of the glass elements. Already during the design stage of the structure the engineer must specify to what extent potential tolerances can be accepted. E.g. the German Standards DIN 18201, 18202 and 18203 part 1 to 3 define the maximum allowable tolerances during construction of the structures and structural elements respectively. Especially in glass design the compliance with the demanded tolerances is a major requirement because stresses and strains resulting from a rigid restraint might lead to a failure of the system. Specifications in regard to the material dependent tolerances in consequence of the production process are regulated in the appropriate standards for building materials. [1, 2] However, the conditions of support, the required width of the joints, and the tolerances as a result of the element fabrication are so far nowhere specified.

In addition to these material and fabrication dependent tolerances certain intrinsic tolerances result from time-dependent deformations of the structure, as there are e.g. creeping and shrinkage, elastic and plastic deflections as well as deformations in consequence of temperature and humidity exposure. The expected tolerances resulting and the subsequent deformations have to be considered also during the design stage.

To determine the actual quality and quantity of the afore stated variations of glass structures, tolerances in consequence of either the production process or the structural work are determined and evaluated by means of statistical methods [3]. Some examples are presented within the presentation. FE analyses and experimental investigations are used to determine how tolerances influence the load carrying behaviour of the structure. Variations of e.g. the dimensions of the glass elements or the positions of the drillings which might result in a rigid restraint are thereby considered. Additionally, possible warping effects resulting in a buckling failure of the elements are investigated. The results from this study will be incorporated in the safety concepts which govern the design and dimensioning of glass structures.

[1] Hansjochen Bludau, Ralf Ertl, Dieter Weber: Maßgerechtes Bauen - Toleranzen im Hochbau, Verlagsgesellschaft Rudolf Müller, Köln 1998.

[2] Denise Eberspächer: Toleranzen und Zwängungen im konstruktiven Glasbau; Festschrift Konrad Bergmeister; Institutsveröffentlichung Heft Nr. 55; ISSN 1028-5334; Wien 2003

[3] Konrad Bergmeister, Denise Eberspächer: Maßtoleranzen nach der Fertigung bei der Firma Eckelt, Bericht Nr. 5 - 03, Wien Juli/August 2003.

Simulation of the fracture behaviour of concrete using continuum damage models at the mesoscale

Stefan ECKARDT, *Institute of Structural Mechanics, Weimar, Germany*

The damage behaviour of heterogenous materials, such as concrete, is generally simulated at the specimen length scale. Continuum damage models with homogenised material parameters are used to describe the material behaviour at the macroscale. The direct determination of the properties of this idealised material model is generally difficult. Furthermore it is not possible to represent all physical effects in the microstructure by macroscale models, e.g. grain decohesion.

At the mesoscale the material structure is described explicitly. As a result the material behaviour can be described for every constituent separately. The simulation of a macroscopic structure at the mesoscale results in models with many degrees of freedom. Hence it is more efficient to use an adaptive heterogenous multiscale approach, which combines the meso- and the macroscale into one numerical model. Only the damaged regions of the structure are simulated at the mesoscale, whereas the undamaged parts are modelled at the macroscale. Based on an macroscopic damage indicator, regions which are initially defined at the macroscale, are transfered to the mesoscale during the loading process.

The emphasis of this presentation is laid on the geometrical description and numerical simulation of concrete at the mesoscale. By modelling the mortar matrix and the aggregates separately, the typical grain structure of concrete arises. The aggregates are generated in accordance to realistic grading curves using ellipsoids. At first, all aggregates of the considered mineral size classes are generated and sorted by volume. Then they are placed randomly one by one inside the specimen starting with the largest grain. To check if two ellipsoids are separated the algorithm by Wang et al.[1] is used. By dividing the specimen into subregions the computation of one aggregate distribution is accelerated.

For numerical simulations the mesomodel is discretised by finite elements. The boundaries between aggregates and matrix are explicitly represented by the finite element mesh. In a first simplified approach rigid bond between the components and linear elastic behaviour of for the aggregates is assumed. A nine parametric constitutive model for concrete, introduced by Pölling[2], is applied to the mortar matrix. This model uses the Rankine criterion to describe the damage behaviour of concrete in tension. In compression the material law combines plasticity and damage theory and a yield/damage potential of Drucker-Prager-type is introduced. Furthermore the model takes into account crack opening and closing under cyclic loading.

To demonstrate the method a two-dimensional example of an artificial concrete with prescribed particle arrangement is chosen and compared with corresponding experimental results.

Acknowledgement: This research has been supported by the German Research Foundation (DFG) through Sonderforschungsbereich 524.

[1] Wang W., Wang J., Kim M.-S. (2001). "An algebraic condition for the separation of two ellipsoids." Computer Aided Geometric Design 18.

[2] Pölling R. (2000). "Eine praxisnahe, schädigungsorientierte Materialbeschreibung von Stahlbeton für Strukturanalysen." PhD thesis, Ruhr-University Bochum.

Special Fast Discrete Dynamic Model of Slender Cantilever Beam

Petr FRANTÍK, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

Jiří MACUR, *Institute of Computer Aided Engineering and Computer Science, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

The derivation of equations of motion of the special cantilever beam model is presented in the paper. The model is created for experimental research of dynamic behaviour of the harmonically loaded slender elastic beam. Very large displacements problems and stability problems (static and dynamic) were expected.

The model is special, because the special discretization method was used. If we consider very slender beam, then the bending oscillations play dominant role in the beam motion. It is possible to neglect the "normal" and the shear strain of the beam and use the "bending" strain only (there are quotes, because both effects are caused by the normal strain). With this neglection, we can divide the beam into finite number of the physical absolutely rigid segments, which are connected by the hinges with the elastic (or arbitrary another) rotational springs, see fig. 1.

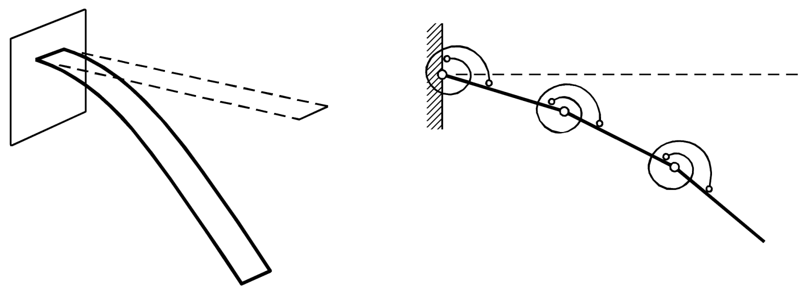


Fig. 1: Real structure and its special model

The neglection of the "normal" and the shear strain causes the increase of the calculations effectiveness and the expressive decrease of the calculations time cost. There is way to fast calculate wide problems with the static and dynamic stability of the beam with arbitrary load and the another nonlinear problems.

The derivation of the model equations of motion give the basic (it means without damping and without any external load) equation system (**??**):

$$\begin{aligned} \mathbf{M}\tfrac{d}{dt}\omega &= \mathbf{Q}\omega^2 - \mathbf{K}\varphi, \\ \tfrac{d}{dt}\varphi &= \omega, \end{aligned} \tag{1}$$

where φ is the vector of the segment rotations, ω is the vector of the segment angular velocities, $\mathbf{M}$ is the full symmetric matrix of the moments of inertia, $\mathbf{Q}$ is the similar full antimetric matrix and $\mathbf{K}$ is the three–diagonal bending stiffness matrix.

The equation system (**??**) of the model can be simple solved by explicite numerical methods (e.g. Euler method, Runge–Kutt method).

Acknowledgement: The research is provided under Czech research project CEZ: J 22/98: 261100009 and supported by grant No. 103/03/0233 from the Grant Agency of the Czech Republic.

Analysis of Thermoplastic Structures

Roman GRATZA, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

Jiří KYTÝR, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

It has been appeared during the use of the thermoplastic structures, that the influence of the time-factor is underestimated as a result of ignorance of material behaviour. Therefore, the natural requirement of the praxis is to analyse the functionality of the plastic structure during its lifetime in more details.

It is necessary to consider the material behaviour with regard to time-dependent factors in the analysis, namely carrying capacity, creep, stress relaxation, shrinkage, swelling and influence of radiation, manufacturing technology, the residual stress, the influence of the environment, coupling method etc. It depends on selection of material and suitability of usage. Subsequently, it is necessary to implicate loading states in the solution, especially change in temperature, magnitude and direction of load, time action, manufacturing phase etc. and to evaluate thermoplastic structures to the adverse reaction of the combination of load states.

Consequently, it is necessary to consider influence of applicability of designing the geometrical shape of the structure, construction details, coupling technology, effect boundary conditions, for example the effect of subsoil etc. It is necessary to deal with physically and geometrically non-linear calculations, stress influence on the structures stiffness, contact problems, stability solution, optimising etc.

Three-dimensional models of the constructions are used for analyses of thermoplastic structures. The ANSYS program system appears convenient for the analysis. It enables to analyse time-depending behaviour of the plastic materials in the calculations.

The thermoplastic structures are evaluated according to some criteria, such as limit conditions of material models, deflections, stability, lifetime, fatigue etc. Generally, time and magnitude load, temperature, influence of environment, style of coupling etc. are considered. The opinion that the plastic materials are convenient only for structures which are loaded adequately and [4]which are used in certain environmental conditions has been thus confirmed.

Acknowledgement: This research has been conducted as a part of the research project CEZ: J 22/98: 261100007.

[1] Gratza, R. and Kytýr, J. Statical Carrying Capacity of Polypropylene Block Structures. In *Proc. of International Conference New Trends in Static and Dynamics of Buildings*, Slovak University of Technology in Bratislava: 2002, p. 69–74.

[2] Gratza, R. and Kytýr, J. The study of linear and nonlinear stability analysis of thermoplastic element. In *Proc. of International Conference New Trends in Static and Dynamics of Buildings*, Slovak University of Technology in Bratislava: 2003, p. 125–128.

A multigrid finite element method for the mesoscale analysis of concrete

Stefan HÄFNER, *Institute of Structural Mechanics, Weimar, Germany*

In classical engineering models concrete structures are usually described by homogenized mean values of material properties as elastic modulus and ultimate stresses. However, concrete, which is mainly consisting of natural aggregates and mortar matrix, is a heterogeneous material with significant physical differences among its components. Therefore even macroscopically applied constant stresses lead to highly nonlinear effective stress distributions on the mesoscopic level. Besides these variations on the load side also the resistance, namely the material strength, changes within a small length scale. Consequently the mechanical behaviour and especially the damage evolution, which is basically a local phenomenon, differ essentially to those of a homogenized material. For most realistic simulation, it is proposed to process a mechanical analysis of concrete on the mesoscale and the presentation describes the first steps towards achieving this concept.

First important aspect is the geometrical modelling of the heterogeneous material. From literature the main characteristics of concrete including particle shapes, particle-size distributions, volume ratios and physical parameters are derived and numerically described. According to predefined parameters, Monte-Carlo-Simulation is used to generate random concrete samples, which statistically meet the designed mixture of components. Besides the general volumetric formula an adequate generation of two-dimensional concrete sections is introduced, which is further applied to numerical analysis.

As any real-size concrete specimen includes numerous particles, the geometrical complexity becomes critical for creating an adequate finite element mesh of inclusion surfaces. Instead of that, uniform element grids are used to modell the structure and the material properties are assigned at the integration points according to the corresponding position of the geometrical modell. The resulting, potentially high number of unknowns raised the need of an efficient solver and as main focal point the development of a corresponding, prototypal multigrid solver is introduced. The algorithm includes mesh adaptivity and cyclic coarse grid correction. Different error influences are considered and assessed for achieving best overall efficiency. Some examples are selected to examine the proposed methods regarding algorithmic aspects, accuracy and computational costs.

[1] Häfner, S., "Effiziente FEM-Multiskalenkonzepte zur Analyse heterogener Materialien", In Proceedings, Forschungskolloquium Baustatik-Baupraxis 2003, September 17-20, Universität der Bundeswehr München–Neubiberg, Germany, 2003

[2] Häfner, S., Eckardt, S. and Könke, C., "A geometrical inclusion-matrix model for the finite element analysis of concrete at multiple scales", In Proceedings, 16th International Conference IKM 2003, Weimar, Germany, June 10-12, 2003. Bauhaus-University Weimar, 2003

Life-Cycle-Oriented Reliability-Based Optimum Design of Structures

Shoko HIGUCHI, *Institute of Structural Mechanics, Bauhaus-University Weimar, Germany*

Optimal economic design of structures requires the minimization of all costs accrued during the entire life-cycle. Thereto structures have to be designed in such a way, that the benefit gained from assuring usefulness, structural integrity and harmlessness to life and limb surpass the expenses spent on construction, maintenance, operation and, finally, demolition. This life-cycle approach to minimization of expenses or maximization of benefits becomes even more mandatory in case of deteriorating structures, where maintenance and repair strategies can be effective ways for delay of aging or wear-out and, therewith, failure or uselessness, but whose associated costs can also amount to a considerable part of the life-cycle cost and therefore can not be justified unlimitedly. From the above follows that an optimal economic design has to take into account simultaneously different optimization criteria like, e.g., sufficient serviceability and reliability against ultimate failure. Although there exist a few approaches with respect to multi-criteria optimization [1], these formulations are based on reliability constraints only, thereby not taking into account the associated costs.

This paper proposes a framework for the optimal design of structures based on the expected total life-cycle cost under reliability constraints of serviceability and ultimate failure. The framework is exemplified for a truss structure subjected to random loading. The reliability constraints of this example are formulated with respect to

1. serviceability, i.e., deformation limits,
2. partial failure of components and
3. ultimate failure of the entire truss structure.

Costs for construction, failure, repair, uselessness, i.e., the violation of serviceability criteria, and operation are taken into account. Design variables are the cross-sectional areas of the truss members. The sensitivity of the optimal design solution with respect to the driving cost parameters is examined.

[1] Frangopol, D. M. (1985). "Structural Optimization Using Reliability Concepts." Journal of Structural Engineering, ASCE, 111, 2288–2301.

Geberal Lateral Pressure Theory and Numerical Methods of Solutions

Tomáš KOUDELKA, *Department of Structural Mechanics, Faculty of Civil Engineering, Czech Technical University, Czech Republic*

This paper discusses numerical methods developed for solution lateral earth pressure. Motivation for us was GLPT (General Lateral Pressure Theory) which establishes new constitutive equations for such contact problem. These equations are highly non-linear and thus we had to use some special approaches, which will be described. Also there will be shown example and compared with current usual methods (DPM).

At present time exist many different approaches for solution lateral earth pressures on retaining structures. At practice is usual used solution using Czech Standard Code. This solution is sufficient for simple structures. For the more precisious computation is possible to use the Dependent Pressures Method (DPM). This method already takes into account dependency of the earth pressure size on the structure deformation and displacements. Another possibility is to use non-linear material equation and the Final Element Method. Among the most used in the geomechanics belong Drucker-Prager and Mohr-Coulomb plastic materials. Also widely used is the Cam-Clay material and its modifications. These models grasp quite good the stress states at soils but this contact problem do not model quite well.

During the last years was publicated a lot of papers which try to describe this contact problem better. One of the possible approaches represents GLPT (P.Koudelka 1996,2001). This theory distinguishes six characteristic values of earth pressures. For the active (ea) and passive (ep) earth pressure it distinguishes three values - at rest, extreme and residual. To these six values correspond characteristic values of horizontal displacements when the values of the earth pressures is reached.

For this theory we developed the method of the numerical solution and used it for the solution of the fixed retaining wall. The wall was modelled as the 2D-beam structure and earth pressures were partly simulated by the load, which depends on the structure displacements, and partly were simulated by the springs with variable stiffness depending on the structure displacements.

The result was compared with the another one obtained using DPM. Both methods gave similar values of the horizontal displacements but the GLPT solution gave greater value of the horizontal displacement of the wall top. Taking into account residual values of the earth pressures caused this effect.

[1] Koudelka, P. (1996). "Philosophy of Earth Pressure Analysis by an Advanced Model". (Proc. IS EC7 Towards Implementation, London.)

[2] Koudelka, P. (2001). "Numerical Analysis of a Physical Experiment with Retained Mass by GLPT". (Proc.RC Geotechnical Engineering in Soft Ground, Shanghai)

Homogenization method for the jointed rock mass

Jiří KRČEK, *Czech Technical University in Prague, Faculty of Civil Engineering, Department of Structural Mechanics, Czech Republic*

There exist a number of methods in the literature to estimate the effective parameters of rock masses. The most common methods are the Self-consistent method [2], the Mori-Tanaka method [3] and the Cai-Horii method [1]. These methods can be grouped into two categories: the iterative and non-iterative methods. The Cai-Hori method belongs to the non-iterative methods and will be the subject of this contribution.

The joints are fundamental properties of the rock masses. Behavior of the rock masses strongly depends on the distribution, orientation and geometry of the joints. The constitutive model in general deals with a wide variety of the distribution of joints. To formulate the stress strain relationship in the jointed rock masses it is necessary to express average stress and strain fields in the representative volume element in terms of the displacement jumps developed along individual joints. The relative displacement jump is evaluate from the joint stiffness and the stress concentration tensor. The stress concentration tensor provides the relationship between the macroscopic stress and the stress acting on the joint. To estimate the stress concentration tensor it is necessary to decomposed the original problem into a homogenous problem, subproblem and a cut-out joint itself. The stress concentration tensor depends on the stiffness of the joint and the system stiffness that includes behavior of other joints and the joint connectivity. The effect of the joint connectivity is included in the reduction of the system stiffness by connectivity coefficients.

Interaction effect of the joints is consider in this model by applying the Cai-Hori method. Following their idea the instantaneous compliance matrix assuming dilute approximation (noninteraction solution) in which a single joint is embedded into an unbounded matrix of the intact rock is determined first. Such a medium is then used as a point of departure for the Cai-Hori method in which each joint is embedded in the effective material whose tangential compliance follows from non-interaction solution. The final overall instantaneous matrix is then arrived at by summing up contributions from all joints. This method unlike the Self-consistent method does not overestimate the interaction effect.

Acknowledgement: Financial support of this work was provided by the research project MSM210000001.

[1] Cai M. and Horii H., A constitutive model of highly jointed rock masses, Applied Mech. Group, Dept. of civil Engineering, University of Tokio, (1991)

[2] Hill R., A self-consistent mechanics of composite materials, J.Mech. Phys. Solids, Vol. 13, 213-222, (1965)

[3] Benveniste Y., A new approach to the application of Mori-Tanaka's theory in composite materials. mechanics of composite materials, Mechanics of Materials, Vol.6, 147-157, (1989)

Lime Plasters with Pozzolanic Admixtures for Historical Buildings: Determination of Basic Thermal and Hygric Properties

Aleš KUNCA, *Department of Structural Mechanics, CTU Prague, Czech Republic*

The lime binder is required for the renewal of plasters used on historical buildings. Lime plasters are damaged more easily than plasters with hydraulic or pozzolanic reactive admixtures. The development of solid plaster structure is quicker in the case of lime-cement mixtures. More suitable and historically used binders are materials based on lime putty and admixtures with pozzolanic properties. In the focus of research are three materials with pozzolana effect admixtures.

- Lime Plaster with Metakaolin
- Lime Plaster with grinded pottery
- Lime Plaster with grinded enamel glass

This material forms first a solid structure of CSH compounds and hydrated calcium aluminates, later calcium carbonates (see [1]-[3] for details). These materials were compared with reference lime plaster.

In the study of the hydrothermal performance basic hygric and thermal storage and transport properties were determined. Comparative measurements for lime plasters were determined, inclusive properties as density and strengths for all plasters. The suitability for newly developed pozzolana plasters for application in reconstruction of historical buildings on the basis of performed measurements is assessed.

Hygric and thermal properties in comparison with lime/sand plaster properties were comparable. Metakaolin reduces density, raises the porosity, reduces thermal conductivity. Sorption isotherms show, that grinded brick potter and grinded enamel glass admixtures does not grow the amount of sorpted water content in the plaster in comparison with the metakaolin pozzolana plaster.

It can be concluded that the most important parameter of the lime-pozzolana plaster studied in this research compared to the lime plaster is a significant increase of both compressive strength and bending strength without using cement that was not used in buildings older than 100 years. At plaster with grinded brick pottery and at plaster with grinded enamel glass were achieved better 28 d compressive strengths than at plaster with metakaolin.

Therefore, all this plasters can be successfully applied in historical buildings instead of the classical lime plaster.

Acknowledgement: This research has been supported by GA CR grant No. 103/02/1081.

[1] Hanykýř, V., Kutzendòrfer, J.: "Technology of Ceramics. Vega", Praha2000.

[2] Takemoto, K., Uchikawa, H.: "Hydration of Pozzolanic Cement. Proceedings of 7th Int. Congress on the Chemistry of Cement, Vol. 1 (Principal reports)" Paris 1980, pp. iv-2/1-29.

[3] Chabannet et al.: "Effectiveness of Metakaolin on the Freezing Resistance of Mortar." Industria Italiana del'Cemento 3, 2001, pp. 240-249.

Material Parameters Identification for Concrete Failure Modeling

David LEHKÝ, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

The utilization of an appropriate material model for realistic modeling of concrete is essential to capture both experimental results and real structures. Generally, the more sophisticated model the more model parameters. In better case basic parameters as compressive strength, modulus of elasticity, etc. are known. In worth case practically nothing is known. Some parameters can be estimated using recommended formulas from literature, but in most cases these formulas can be used only as a first approximation of the parameters. So when experimental load-deflection curve should be captured by e.g. nonlinear fracture mechanics model, first calculation using initial set of material model parameters usually deviates from desired one. Then it is necessary to make some corrections of the parameters using trial-and-error method. Parameters are changed step by step, numerical calculations have to be carried out repeatedly many times and numerical simulation results are compared with experimental results. This classical approach is not very efficient, especially if a complex material model with many parameters is used. That is why different alternatives of identification algorithm have been proposed in literature and they are becoming more and more attractive.

A new approach for material model parameters identification is presented here. The proposed approach is based on coupling of the stochastic nonlinear fracture mechanics analysis and the artificial neural network. Identification parameters play the role of basic random variables with the scatter reflecting the physical range of possible values. The efficient Monte Carlo type simulation method Latin Hypercube Sampling (LHS) is used. This statistical simulation offers the set of data, a "bundle" of numerically simulated load-deflection curves. Generated basic random variables and load-deflection curves are used for training of a suitable type of neural network. Once the network is trained it represents an approximation, which can be utilized in an opposite way: For given experimental load-deflection curve to provide the best possible set of material model parameters.

Several software tools had to be combined in order to make the identification possible. First, nonlinear fracture mechanics software ATENA and probabilistic software package FREET - the combination is represented under the name SARA. Second, new neural network software DLNNET has been developed.

The suggested methodology is demonstrated on selected numerical examples of identification of material model for concrete called SBETA: notched specimen under three-point bending of high-strength concrete used for railway sleepers and reinforced concrete shear wall experiments. Promising good results have been achieved, which indicates that efficient techniques have been combined at all three basic levels: deterministic nonlinear modeling, probabilistic stratified simulation and neural network approximation.

Acknowledgement: Projects GA ČR 103/02/1030 and CEZ: J22/98: 261100007.

[1] Novák, D., Lehký, D., Keršner, Z. (2003). "Lomově–mechanické parametry vysokohodnotného betonu: experiment, modelování a identifikace. (Fracture Mechanics Parameters of High Performance Concrete: Experiment, Modeling and Identification.)" 2. mezinárodní sympozium FC & HPC - Fibre Concrete & High Performance Concrete, Malenovice, ISBN 80-86604-08-X, 2003, pp. 67-74, (in czech).

Search for microplane model parameters

Matěj LEPŠ, Anna KUČEROVÁ, Zdeněk BITTNAR, *Department of Structural Mechanics, Faculty of Civil Engineering, Czech Technical University in Prague*

Through the last decades, hand in hand with the growth of computer performance, the so-called artificial intelligence or soft-computing methods were developed as alternatives to traditional solutions of problems which are difficult to be defined, described or resolved using traditional ones. Genetic algorithms (GA's) [1] are good examples of these methods. Some improvements during our previous work on GA resulted in the SADE [2] algorithm.

This optimization tool is hereafter used to find a set of parameters of a constitutive model for concrete called the microplane model [3]. It is a fully three-dimensional model, which includes many types of loading, unloading and cyclic loading and incorporates the development of anisotropy within the material. The main disadvantage of this model is the numerical demand, which is manifold in comparison with traditional approaches. For the microplane model, a particular type of concrete is described using 8 parameters: Young's modulus, Poisson ratio and other six phenomenological parameters, which even do not have a simple physical interpretation, so it is difficult to determine their values from an experiment. A usual scenario is that an experimenter is trying to fit stress-strain diagrams varying the model parameters by the trial and error method. As one of the most up-to-date approaches to estimation of material parameters, stochastic optimization methods could be employed. The preliminary results of our computations show that the SADE algorithm has the ability to find the microplane model parameters with a satisfactory precision.

The disadvantage of parameter finding are the computational demands related to the structural FEM analysis. Our solution to this obstacle is the parallelization of proposed method. So-called "Global parallel model" proposed in [4] is used. More specifically, the program is divided into an optimization and an analysis part and in this way it is implemented in the cluster of PCs.

Results of this project show that, in the engineering practice, an application of automated optimization procedure could save a huge amount of an experimenter's time needed for searching parameters of the microplane material law, or even more sophisticated model, by the trial and error method. In addition, the obtained numerical results show that the computational demand of this task ensures the nearly-linear speedup even for relatively high number of processors.

Acknowledgement: The authors would like to acknowledge the financial support of this work by research project of Ministry of Education, MSM 210000003.

[1] Z. Michalewicz. (1999) "Genetic Algorithms + Data Structures = Evolution Programs". (Springer-Verlag).

[2] O. Hrstka, A. Kučerová, M. Lepš and J. Zeman. (2001) "A competitive comparison of different types of evolutionary algorithms". (In Proceedings of the Sixth International Conference of Artificial Intelligence to Civil and Structural Engineering, Civil-Comp Press).

[3] J. Němeček. (2000) "Modelling of compressive softening of concrete". (CTU Reports, 4(6), Ph.D. Thesis).

[4] E. Cantú-Paz. (2001) "Efficient and Accurate Parallel Genetic Algorithms". (Kluwer Academic Publishers).

Dynamic Response of Structure in Fluid Flow

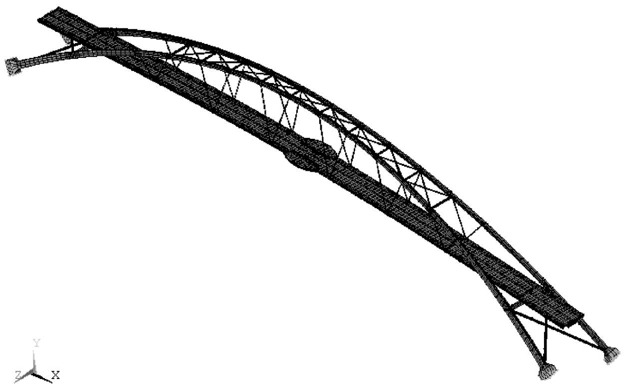
Václav LOSÍK, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

Structures are located in the open exterior where they are subjected to influence of a wind flow. Naturally, we can imagine that there is a static action of the wind which applies pressure to the structure. But when some conditions are met, the periodic fluctuations are developed behind the structure and begin to dynamicly actuate the structure which reacts by dynamic response.

This dynamic response occures mainly on structers which have one dimension highly exceeding the others eg. like towers, bridges or submerged floating tunnels (vision). When some big structure is supposed to be constructed there is a need to examine behavior of the structure. Diminished model is usually built and its response is analysed in a wind tunnel. This method allowes to investigate only static response of the structure. But because of a size effect it is impossible to analyze mentioned dynamic response. And because of that, there is a necessity to find another method to examine the structure.

In case we expect the dynamic movement of the structure small in comparsion with dimensions it is possible to omit it. In that case the problem is dividet into two separated parts. The first part CFD analysis can provide a time history of the wind pressures, in the other part the pressures are applied on a finite-element mechanical model and the dynamic response has been solved.

The problem of dynamic response of a light foot bridge with the span of 100 meters which was just described is solved in this work. 2D model was built for CFD analysis with a cross section of the bridge. Pressures were solved and subsequently applied as a load in harmonic analysis which was performed on detailed 3D mechanical model of the bridge.



Acknowledgement: This research has been conducted as a part of the research project CEZ: J 22/98 261100007.

Utilization of the Natural Neighbour Interpolation to simulate random fracture of concrete

Thomas MOST, *Institute of Structural Mechanics, Weimar, Germany*

In the recent years meshless methods have become very popular for the simulation of discrete crack propagation. This is caused by the fact, that meshless shape functions only depend on the node positions in opposite to these of finite elements, where a predefined element mesh is necessary. Through the continous character of the meshless shape functions, the most methods show C^1 continuity almost everywhere, an uncomplicated transformation of history variables is possible, which is necessray by adaptive strategies.

The most common meshless interpolation scheme is the "Moving Least Squares"(MLS) procedure, which is used in the "Elementfree Galerkin Method". Through the approximation character the interpolation function does not pass through the nodal values. The essential boundary conditions are not fullfilled automatically, which complicates the coupling with finite elements. Because of this fact an alternative method will be presented here, which uses the neigborhood relations of the nodes, the so-called "Natural Neighbor Interpolation". The mechanical implementation is called "Natural Element Method"(NEM) [1]. By this method the computation of the shape functions is based on a Voronoi diagram of the domain which is the dual of the Delaunay triangulation. The interpolation procedure fulfills the boundary conditions automatically. It enables the representation of discontinuities analogous to the MLS-interpolation.

For the modelling of concrete fracture a "Cohesive Crack Model"[2] is used, where a force transmission over the crack surface depending on the crack width is introduced. Because of this fact, we do not get the singularity field at the crack tip as by stress-free cracks. Therefore the Rankine criterion is used instead of the concept of stress intensity factors for crack initiation and crack growth, whereby a nonlocal stress computation is included. The crack growth is done incremental, the algorithm, which controls the crack increment length is presented in detail.

Because of the variation of the material properties of concrete a stochastic modelling in form of random fields is applied. Thereby the spectral representation of the random field is used for the easy computation of discrete samples. To determine the statistical properties of the nonlinear stochastic response of an investigated structure, improved "Latin Hypercube Sampling"[3] is applied. The number of required samples is much less than by using Plain Monte Carlo Simulation, which leeds to a remarkable reduction of the numerical effort. The presented crack modelling algorithm is verified on some selected deterministic and stochastic numerical examples.

Acknowledgement: Supported by the German Research Council (DFG) through *Collaborative Research Center 524* .

[1] N. Sukumar, B. Moran, and T. Belytschko. "The Natural Element Method in solid mechanics." *International Journal of Numerical Methods in Engineering*, 43:839–887, 1998.

[2] M. Hillerborg, M. Modeer, and P.E. Peterson. "Analysis of crack formation and crack growth in concrete by means of fracture mechanics and finite elements." *Cement and Concrete Research*, 6:773–782, 1976.

[3] D. Novák et. al. "Simulation of random fields based on orthogonal transformations of covariance matrix and latin hypercube sampling." *Proc. Int. Conf. on Monte Carlo Simulation*, Swets&Zeitlinger, 2001.

Flow-Induced Vibrations of Slender Line-Like Structures

Aleš NEVAŘIL, *Institute of Structural Mechanics Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

This paper deals with structural constructions in the Earth's atmospheric field. Among these structures belong slender line-like constructions of towers, guyed masts, transmitters, some of cable-anchored structures, etc.

These structures are loaded by the wind, causing on them the following forces:

- Aerodynamic forces (not dependent on the movement of the structure) – mostly produced by longitudinal effect of the turbulent wind. These forces relay on dynamic properties of the structure, i.e. its natural frequencies, damping ratios, eigenmodes.
- Aeroelastic forces (dependent on the movement of the structure) – longitudinal and also transversal motion of the structure (twisting movement is also possible) is produced by the motion of the structure in the wind flow.

The interaction of the solid structure and the fluid wind field (FSI) produces several types of responses:

- Stochastic response – typical for turbulent wind flow with high turbulence intensity. The resultant of the pressure generates dragging force, lifting force and torque moment.
- Vortex induced vibrations – are produced in the transverse direction in the case of an adequately flexible structure and natural frequency close to vortex shedding frequency. It is also referred to as von Kármán effect or so-called lock-in phenomenon.
- Galloping – are transverse vibrations of the structure imposed by non-symmetric vortex path. It may be developed in case of wind flowing across non-symmetric cross-section shapes. Cylinders may become sensitive to galloping when iced (freezing rain conditions) or in conditions of wind and rain, when it is problematic to examine this phenomenon.
- Torsion divergence – is an aerodynamic instability developed by the combination of drag and torsion components of the wind load. It may be analysed as a static phenomenon.
- Flutter – is an aerodynamic phenomenon produced by a combination of lift and torsion component of the wind loading. It is a dynamic phenomenon.
- Other responses – interference galloping, wake galloping, buffeting, ovalling, etc.

A number of research projects have already been done in the field of the FSI investigation, but yet some questions remain answered not clearly enough. Among these belong the following ones:

- It is difficult to derive aerodynamic coefficients values based upon the mathematical model of the media flow for the cross-section shape influenced by the flutter phenomenon.
- The selection of suitable spectral density values of a modal load using statistical methods in the calculation is a primary problem of the buffeting phenomenon analysis.

Acknowledgement: This research has been conducted as a part of the research project CEZ: J 22/98: 261100007.

Modelling of the Charles Bridge Masonry Wall on Meso Scale with Commercial Software

Jan NOVÁK, *Faculty of Civil Engineering, Czech Technical University, Czech Republic*

On meso-scale the masonry walls can be considered as a periodic two-phase composite material. The mechanical response of such a system then depends on the behavior of each of the two phases, brick and mortar. To estimate the homogenized or equivalent material properties of the masonry structures it is possible to the exploit well known techniques known from the micromechanical analysis of classical composites [1]. Among others, the analysis based on unit cell computations appears as the most appealing one.

The main objective of this contribution is to address this problem from the implementation point of view. While formulating the basic equations such as those proposed in [2] is relatively simple, their implementation into a general purpose Finite Element Codes, e.g. ANSYS, may become a difficult task. In this regard, the key point is an introduction of the periodic boundary conditions and loading the periodic unit cell such that it produces uniform macroscopic fields.

When searching for the effective elastic stiffness the required load after imposing the multi-point constraints equivalent to the periodic boundary conditions can be applied by prescribing displacements to selected corner nodes of the periodic unit cells. Under plane stress assumption the effective stiffness matrix follows from three independent unit load cases that correspond to loading the unit cell, in turn, by each of the three components of the macroscopic strain vector, while the other two vanish. The volume stress averages than supply individual entries of the desired effective (macroscopic) stiffness matrix. In analogy, one may enforce the traction boundary conditions to arrive at the effective compliance matrix.

An application of this approach combined with the ANSYS Finite Element program is demonstrated by homogenizing a selected unit cell of the Charles Bridge masonry.

Acknowledgement: Financial support for this project was provided by the Ministry of education, projects No. MSM:210000001 and MSM:210000003.

[1] Sejnoha M., Zeman J., "Micromechanical Analysis of Random Composites", CTU Reports, Vol. 6, 2002

[2] Michel, J.C. and Moulinec, H. and Suquet, P., "Effective properties of composite materials with periodic microstructure: A computational approach", Computer Methods in Applied Mechanics and Engineering, Vol. 182, 109–143, 1999

Modelling of Anchorage Blocks Tests

Florentina PERNICA, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

The work is focused on numerical computational modeling of experimental results of anchorage blocks with CFRP strips. The aim of the experimental work performed at Faculty of Civil Engineering, Department of Concrete and Masonry Structures, was to determine the influence of anchorage length on the ultimate force on the strip (Švaříčková 2003). The idea to capture the experimental results has been established in order to make a consequent virtual computer testing of the tested specimen. For modeling, nonlinear fracture mechanics software ATENA 2D was used as it enables an efficient capturing of both pre-peak and post-peak behavior.
Experiment: The anchorage blocks of dimension 150/150/450 mm were prepared from concrete of class B15-B25. The physical and mechanical properties of concrete specimens were determined by non-destructive methods before the beginning of the tests. The CFRP strips of cross-sectional dimension 50/1,2 mm and Young's modulus 155 Gpa was bonded to the prepared surface of different anchorage lengths - 150, 225 to 300 mm. The resistance strain gauges were glued on the strips and the deformation along the length of anchorage was measured with video-extensometer. The arrangement of the test was: anchoring without stirrup, anchorage with stirrup without pressure and anchoring with stressing stirrups. The outcome of experiments was diagrams of strain profile along the length of CFRP strip. Unfortunately, no load-deflection curve has been monitored, which made numerical modeling more complicated.
Modelling: The mathematical model was performed using software ATENA 2D that was developed mainly for the simulation of real behavior of RC structures. The software ATENA represent the nonlinear fracture mechanics FEM tool for realistic modeling of cracking in quasi-brittle materials. FEM 2D ATENA computational model has been developed in several test alternatives. For modeling of concrete and adhesive SBETA material model was used and plane strain elastic isotropic material was selected for CFRP modeling. In the mathematical model monitors have been placed in the points where the tension meters were glued. The monitors scanned strips displacement in the force direction. For data evaluation were created cuts in monitoring points. After calculation from this cuts the values of strain of strip were taken.
The idea was to compare the diagram – profile of strain of strip along carbon fiber, from the experiment with the graph obtained from the ATENA computational model. The trends of strain profile curves of both experimental and numerical results were very similar, but an absolute value was rather different due to insufficient previous material parameters calibration. A consequent study can continue – a virtual experiments using computer simulation of next cases can be based on these calibrated models.

Acknowledgement: Projects GA ČR 103/02/1030 and CEZ: J22/98: 261100007.

[1] Červenka, V. & Pukl, R. (2003). "ATENA Program Documentation." Červenka Consulting, Praha.

[2] Švaříčková, I. (2003). "Problematika zesilování stavebních konstrukcí externí lepenou výztuží z uhlíkových vláken", Ph.D.Thesis, Superviser: Prof. RNDr. Ing. Petr Štěpánek, CSc., Faculta Stavební, VUT Brno,(in print).

Characterization of constraint effect and estimation of T-stress by finite element method

Stanislav SEITL, *Institute of Physics of Materials, Academy of Sciences of the Czech Republic, Brno*

Knowledge of the corresponding values of stress intensity factor K [3] and T-stress is the primary assumption for successful application of two-parameter linear elastic fracture mechanics in the technical practice.

$$\sigma_{ij} = \frac{K_I}{\sqrt{2\pi r}} f_{ij}(\varphi) + T\delta_{1i}\delta_{j1} \qquad (1)$$

The T-stress, which can be considered as a stress parallel to the crack flanks, can be either tensile or compressive. In a case of simple and frequently used bodies (e.g. Compact tension specimen-CT, Center Cracked plate tension specimen-CCT and so on) the corresponding values can be found in the literature dealing with two-parameter fracture mechanics. In complicated cases one of the existing numerical methods has to be used.

All the analysis performed in this contribution were carried out using the finite element method under the assumption of elastic material behavior. Calculations were modelled using the finite element package ANSYS. The commonly used procedures for calculation of T-stress values by finite element method, namely the direct [1], [4] and the integral [2] approaches, are presented and discussed in this contribution.

The aim of the contribution is to provide a survey of these procedures and to point to their advantages and disadvantage. Contribution is appended with practical examples of T-stress calculation.

Acknowledgement: This work was supported by the grand No. 101/03/0331 from Grant Academy of the Czech republic.

[1] Bednar, K. (1999). "Two-parameter fracture mechanics: calculation of the relevant parameters and investigation of their influence on the fatigue crack propagation", Ph.D. thesis, Brno, (in Czech).

[2] Groger, R. (2001). "An analysis of algorithms for the determination of T stress with respect to the higher-order terms of the Williams eigenfunction expansion", Proceedings of seminar Problems of Fracture Mechanics I., Brno.

[3] Knesl, Y., Hutar, P., Seitl, S. (2002). "Calculation of stress intensity factor by finite element method", Vypocty konstrukci metodou konecnych prvku, Praha, (in Czech).

[4] Yang, B., Ravi-Chandar, K. (1999). "Evaluation of elastic T-stress by the stress difference method", Engineering Fracture Mechanics, Volume 64.

Probabilistic modeling as a base for the life time planning

Alfred STRAUSS, *Wien BOKU*

The general approach for safety evaluation of existing bridges is based on codes and different specific regulations. It has been found that reliability assessment which is going beyond the boundaries of codes can bring a significant money saving and provide a new insight into bridge administration and decision-making process, Casas et al. (2002), Enevoldsen (2001), Frangopol (2000). Latest developments show that the optimum balance between cost and safety of concrete structures, e.g. bridges, is becoming a common problem world-wide. Methodologies for use of probabilistic based assessment are available and have been proven to work in practice. But suitable tools for use in design offices are generally missing. This contribution shows the combination of efficient techniques, fracture mechanics, monitoring and reliability engineering to achieve the goal: to assess realistic behavior of concrete structures from the reliability point of view. The aim is to present a procedure that allows the probabilistic-based assessment of structures, which can be used in combination with degradation-models and monitoring data for a efficient life cycling planning.

Probabilistic based assessment: The procedure for the probabilistic based assessment is based on a recently developed integrated system of nonlinear fracture mechanics software ATENA and probabilistic module FREET called SARA (Structural Analysis and Reliability Assessment). The Latin Hypercube Sampling technique, used in this integrated system, which requires rather small number of samples for accurate results as opposed for the statistical Monte Carlo simulation shows an efficient technique to solve nonlinear analysis which is computationally intensive. The presented approach has been applied recently for several reliability problems, Bergmeister et al. (2002), Novák et al. (2002), Pukl et al. (2002, 2003). This approach was already used in several problem definitions of the engineering nature and gives an deeper insight due to the included sensitivity analysis in the structural behaviour.

Degradation-Models: A structure is subjected to degradation within its lifetime and therefore will lose at reliability. A suitable tool in combination with SARA are Weibull-functions to describe the degradation behavior of stuctures over their lifetime. The information, get from every inspection, can be included into the Weibull-parameters. A continuously updating of the degradation line after each inspection therefore is possible. These degradation-lines are decision tools for the preservation and therefore also for the lifetime planning.

Safety Index: The generall reliability considerations in engineering refer to a safety index β of 4.70 for a reference time of one year. These demands are formulated in new stuctural codes and refer to new structures based on current social needs.However: Is it also possible that the safety index - safety standard is subjected to fluctuations or other influences over a longer time period? and Is it necessary for existing structures to demand the safety features than for new ones? Regarding this thematic area an approach for required safety indexes worked out by Strauss (2003)will be presented.

[1] Casas, J.R., Frangopol, D.M. and Nowak, A.S. 2002.Bridge Maintenance, Safety and Management. Proceedings of the First International Conference IABMAS'02, Barcelona, Spain: CIMNE.

[2] Enevoldsen, I. 2001. Experience with probabilistic-based as-sessment of bridges. Structural Engineering International 11 (4): 251-260.

[3] Frangopol, D.M. 2000. Advances in life-cycle reliability-based technology for design and maintenance of structural sys-tems. In: B.H.V. Topping (ed.) Computational mechanics for the twenty-first century. Edinburgh, UK, Saxe-Coburg Publications: 311-328.

[4] Bergemeister, K., Strauss, A., Eichinger, E.M., Kolleger, J., Novák, D., Pukl, R. and Cervenka, V. 2002. Struct. analysis and safety assessment of existing concrete structures. In: The first fib Congress, Osaka, Sess. 11: 47-54.

[5] Novák, D., Pukl, R., Vorechovsk, M., Rusina, R. and Cervenka, V. 2002. Structural reliability assessment of computationally intensive problems - non-linear FEM analysis. In: ASRANet Colloquium, Glasgow, Scotland.

[6] Pukl, R., Novák, D. and Eichinger, E.M. 2002. Stochastic Nonlinear Fracture Analysis. In: 1st International Confer-ence on Bridge Maintenance, Safety and Management (IABMAS'02), Barcelona, Spain.

[7] Pukl, R., Novák, D., and Bergmeister, K. 2003. Reliability as-sessment of concrete structures. In: Computational model-ling of concrete structures (Euro-C 2003), St. Johann in Pongau, Austria.

Modeling the behavior of concrete exposed to high temperatures

Jiří SUROVEC, *Department of Structural Mechanics, Faculty of Civil Engineering, Czech Technical University in Prague, Czech Republic*

Truly ideal material model for concrete that accurately represents behavior of concrete heated to elevated temperatures caused by fire should take into account more than only the standard mechanical properties. The damage invoked by fire load depends on many factors. From this point of view a necessity of complex model has to be solved. It should simultaneously include non-linear transport of water, water vapor, air and heat as well as fracture of concrete. These processes affect internal pressure in concrete pores, phase change of water to gas state or chemical changes in concrete such as dehydration and calcination process in aggregate. Excessive increment of internal pore pressure can cause brittle spalling (that is a brittle failure with most cracks parallel to the heated surface) and uncover the reinforcement. The summary of processes taking place in heated concrete such as liquid water flux due to pressure gradient, chemical processes or spalling will be described.

The numerous thermal processes may render the use of the general models too difficult for engineering practice. The main disadvantage of such a model is lack of accurate values for numerous initial material constants or boundary conditions. It is difficult to predict the rate of change of material parameters. However, usually only a direct compression or direct tension tests with controlled deformation are available. The importance of thermo-physical processes included will be shown and omitting of less important processes will be proposed.

Our aim is to make the model as simple as possible and to include this model to a commercial FEM software package ATENA. The relations will be formulated phenomenologically. The dependence of mechanical properties such as modulus of elasticity, compression and tension strengths, ductility and fracture energy both in tension and compression on temperature will be presented for different types of concrete.

At the very first stage of this project the existing damage model included in ATENA was used with mechanical properties dependent on temperature. The uncoupled problem was solved. At first the thermal analysis was performed, which provided time-dependent distribution of temperature throughout the structure. Then a different material properties of each element were distributed according to actual element temperature. Then a non-linear fracture analysis was performed. The results of validation tests will be shown and the use of such a simple model will be discussed.

Second stage of our project includes strictly mathematical solution using Mathematica software. Complex model with both material and energy transport was used. Simple one dimensional bar analysis has been performed to show the mutual dependencies of internal processes. The model was formulated as a set of coupled partial differential equations which were directly solved. The aim is to find relations between the modeling complexity and importance of each material process; to allow consideration of engineering simplifications. The results will be presented. The influence of different initial conditions will be discussed.

In the final summary, the ways to continue our research will be proposed. The possibility of application of our future material model to engineering practice with special attention to new materials used in tunnel engineering, such as high strength concrete, polymeric fiber reinforced concrete, etc. will be presented.

Acknowledgement: Financial support of this work was provided by the EU UPTUN research project contract number G1RD-CT-2002-00766.

[1] Tenchev, R.T., and Li, L.Y. and Purkiss, J.A., "Finite element analysis of coupled heat and moisture transfer in concrete subjected to fire". *Numerical heat transfer Part A-Aplications*, 39(7), May 25 2001, pp. 685–710.

[2] Felicetti, R., Gambarova, P.G., "Effects of high temperature on the residual compressive strength of high-strength siliceous concretes", *ACI Materials Journal*, 95(4), July-August, 1998, pp. 395–406.

Adaptivity in nonlinear problems of elasto-plasticity

Ladislav SVOBODA, *Department of Structural Mechanics, Faculty of Civil Engineering, Czech Technical University, Czech Republic*

Recently the author have discussed error estimation procedures and adaptive mesh design based on recovery of stresses, strains and other gradients for linear problems. Application of such methods to practical engineering analysis is widely available today. Presently we are engaged in the extension of the methodology to nonlinear problems. At first glance, it appears simple because the solution is generally obtained by successive linearization, however several difficulties arise which we shall discuss here. The applications in this contribution are focused on static problems of elasto-plasticity, common in engineering.

Generally, due to path dependency of the solution in materially nonlinear analyses, the loading is applied in an incremental manner with an iterative linearization, using the Arc-length method in each increment. The iteration is usually continued until a suitable norm of variables falls under a prescribed tolerance. Depending on the tolerance used in the termination of Arc-length iteration, an additional error to that due to discretization will occur in the computational process. This contribution mainly focuses on the discretization error by assuming that the magnitudes of the increments and tolerance used are sufficiently small to make the iteration errors secondary.

The observed error is function of the loading and the decision of remeshing, while the nonlinear computation is being carried out, requires important engineering judgement. An appropriate norm of variables must be devised to monitor the accuracy of the solution helping to recognize the necessity of remeshing.

Various measures of error are adapted by investigators (the L_2 norm of displacements, the energy norm or local indicators). In this contribution we will use energy norm computed from recovered stresses, which have proved to be the best in linear problems. The norm will be slightly adapted to suit plasticity problems. Obviously, success of this error estimation is depended on exactness of recovery method used and therefore we shall take the advantage of SPR (Superconvergent Patch Recovery) method, which has shown excellent performance in linear problems.

The majority of investigators computes error on the whole domain, it means also in plastic zones. Therefore they use incremental energy norm. In n-th increment the corresponding error is computed and added to the total error reached in increment $n-1$. The total error is stored at integration points and after generating a new mesh, corresponding values must be constructed for the new integration points. In this contribution, however, the accuracy is controlled in elastic zones only. Therefore, in every increment the total error is computed again, thus it is not necessary to store the values. Then the error is compared with the prescribed limit, which is not, however, constant over the domain. The highest accuracy is required on elements close to plastic zones.

In this contribution, after a short overview of general formulation of plastic problems, we explain the error norm used in the error estimator. The problem of updating the state variables after remeshing is discussed in some details too. Finally, we conclude our contribution by some numerical examples.

Acknowledgement: Financial support was provided by IGS grant No. 0301011.

Nonlinear dynamics calculations in structure FEM-solvers

Zbyněk VLK, *FEM Consulting/Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

Structure solvers that are prepared by company FEM Consulting. s.r.o (FEM-Solver) is used in programs for static analysis of structure models (for example Nexis32, Rfem). The calculations of nonlinear dynamics models in FEM-Solver are based on the current solving static nonlinear problems by Newton-Raphson method. The paper deals with combining Newton-Raphson method with Newmark method that is used for time analysis.

Fundamental equation of motion

$$\mathbf{M} \cdot \mathbf{\Delta a}_i + \mathbf{C} \cdot \mathbf{\Delta v}_i + \mathbf{K} \cdot \mathbf{\Delta u}_i = \mathbf{\Delta F}_i \quad (1)$$

is possible to transform to equation

$$\mathbf{\Delta F}_{S,i} = \mathbf{\Delta F}_i - \mathbf{\Delta F}_{D,i} \quad (2)$$

where

$\mathbf{\Delta F}_{D,i} = \mathbf{M} \cdot \mathbf{\Delta a}_i + \mathbf{C} \cdot \mathbf{\Delta v}_i$,
$\mathbf{\Delta F}_{S,i} = \mathbf{K} \cdot \mathbf{\Delta u}_i$ for linear analysis and
$\mathbf{\Delta F}_{S,i} = f(\mathbf{\Delta u}_i, \mathbf{\Delta u}_{i-1}, \mathbf{\Delta F}_{S,i-1})$ for nonlinear analysis.

For calculation of increment of deformations $\mathbf{\Delta u}_i$ from vectors $\mathbf{\Delta F}_{S,i}$ is used FEM-Solver. A sequence of calculation illustrates the next figure.

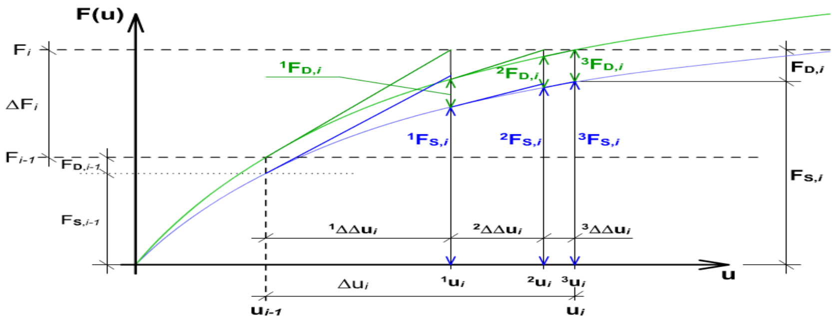


Fig. 1

Acknowledgement: The research is provided under Czech research project CEZ: J 22/98: 261 100 007.

Application of Sparse Direct Solver on Large Systems of Linear Equations

Richard VONDRÁČEK, *Department of Structural Mechanics, Faculty of Civil Engineering, Czech Technical University in Prague, Czech Republic*

Solution of large system of linear equations can be obtained by direct methods or by iterative methods. Direct methods, usually based on various matrix decompostions, are rather robust for wide breadth of matrices arising from different numerical applications but sparsity pattern and the fill-in of the characteristic matrix is the issue which determines whether the problem is solveable in the scope of given time and memory limits or not.

Iterative methods allow solution of much larger problems, because iterative methods can fully exploit the sparsity of the characteristic matrix. The drawback of iterative methods is that the number of iterations increases with the growing condition number of the matrix. Mechanical problems of plate-shell structures are typical examples where the iterative methods tend to be inappropriate.

Modern computational approaches are based on domain decomposition methods. First merit of domain decompostion methods is their natural suitability for parallel processing. The other benefit of domain decomposition methods is the practical decoupling of the problem into local and global layer where each of these layers can be solved appropriately by direct or iterative method.

The contribution will present principles and application of Schur complements method (primal domain decomposition), FETI method (dual domain decomposition) [2], DP-FETI method (dual-primal domain decomposition) [3].

All above stated methods require efficient implementation of the direct solver. This contribution introduces direct solver based on sparse $\boldsymbol{LDL}^T$ decompostion. The matrix is stored by the compressed row storage and the sparsity of the final matrix $\boldsymbol{L}$ is secured by initial approximate minimum degree ordering algorithm [1]. The sparse direct solver based on compressed column storage is compared with direct solver based on skyline storage of the matrix.

The presentation will briefly summarize the principles of the three domain decomposition methods and show the ideas behind the implementation of the sparse direct solver. Finally, some numerical examples of the methods with corresponding benchmarks will be presented.

Acknowledgement: Projects GAČR 103/01/0400 and Czech Academy of Sciences S3086102.

[1] Davis T.A., Amestoy P., Duff I.S. (1994). "An Approximate Minimum Degree Ordering Algorithm." Computer and Information Sciences Dept.,University of Florida, Technical Report TR-94-039

[2] Farhat, Ch., Roux, F.X. (1991) "A Method of Finite Element Tearing and Interconnecting and its Parallel Solution Algorithm" International Journal for Numerical Methods in Engineering, 32, 1205-1227,

[3] Farhat C., Lesoinne M., LeTallec P., Pierson K., Rixen D. (2001) "FETI-DP: A Dual-Primal Unified FETI Method–Part I: A Faster Alternative to the Two-level FETI Method." International Journal for Numerical Methods in Engineering; 50(7): 1523-1544

Dynamic response of a bridge subjected to moving vehicles

Petr VYMLÁTIL, *Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic*

Traffic loads have a great effect on dynamic stresses in bridge structures, and cause them to vibrate intensively, especially at high velocities. The determination of the dynamics effect of moving loads on elastic structures particularly on bridges is a very complicated problem. This is because the dynamic effects induced by moving vehicles on the bridge are greatly influenced by the interaction between the vehicles and the bridge structure. The large-span bridges have a mass much larger then the mass of the vehicle, and very low first natural frequencies. If vibration of vehicles on own springs is neglected, the effect of moving vehicles may approximately be replaced by the effects of moving forces. The objective of this study was to analyse the slender steel bridge dynamic response using a simple FEM bridge model and simple vehicle model reduced to the system of forces. To solve this problem the FEM program system ANSYS is used. Obtained results are compared with experimental results obtained in the course of dynamic loading test of the bridge.

Object of the analysis is a road bridge over the Labe river. It is the tree-span steel bridge structure with a composite road deck. Span lengths are 36, 128 and 36 m. The flexible road deck is in the mid span stiffened with two parabolic arches and suspended on the hangers.

The FEM-based model have to be very simple due computational expenses. For modelling all girders, arches and hangers are used the beam elements, for modelling the concrete road deck are used the shell elements. Geometric and material linearity is considered.

For evaluating the traffic vibration response is used the Newmark time integration method with the time step 0,005 s. The symmetrical traverse of two automotive trucks is simulated. The solution is performed with different travelling speeds. In initial approach constant intensity of axial loads is assumed. The calculated response is in good agreement with measurement. Difference between solution and measurement may be ascribed to ignore inertial effect of moving mass on the deformed structure and surface roughness. These flaws can be eliminated by using axial loads with harmonic component. The frequency and the amplitude of the exciting forces can be determined approximately from measured data.

For more sophisticated analysis of the bridge/vehicle interaction is necessary to model the vehicle like a system with more degrees of freedom and continuously or discretely distributed mass, with correct interaction between the bridge deck and vehicle wheels.

Acknowledgement: This research has been conducted as a part of the research project CEZ: J 22/98 261100007

Development of soft tissue models using finite elements

Benjamin WINKEL, *Institute of Structural Mechanics, Weimar, Germany*

When modelling biological tissue for surgical applications and impact simulations the developer has to deal with large deformations, incompressability, time and activation dependency, and the necessity of high resolution models that pose a challenge to computing powers. An outline is given on the current approach within the sofware package SLang.

The first concern is the implementation of finite elements with the ability to simulate structures undergoing near-isovolumetric large deformations. Traditional displacement based elements do not fulfill the corresponding requirement $sp(\boldsymbol{\epsilon}) = 0$ pointwise as the Poisson ratio approaches 0.5 or the compression modulus approaches ∞ respectively. Hence, even in the absence of volumetric strain remains an internal energy term $\Pi_{vol} \gg \Pi_{dev}$ that causes the so-called volumetric locking. The examination of the stiffness matrix' eigenvalues is used as a conclusive method to judge an elements liability for this locking effect, as inhomogenous dilatational modes are numerically feigned.

Various element formulations have been developed to manage incompressibility, mainly in the context of simulating hyperelastic rubber and elasto-plasticity in metals. The EAS method led to elements that avoided volumetric and shear locking effectively in the linear elastic case, but are subject to hour glass modes under large strains [1]. SRI-, u/p- and $\overline{\mathbf{F}}$-elements are less prone to this undesirable effect. As hour glassing was shown to occur when the inf-sup condition is not satisfied combined formulations have been introduced to optimize the elements behavior in that way. However, even those advanced elements do not reliably prevent hour glassing under large strains [1].

Therefore a trilinear 8-nodal $\overline{\mathbf{F}}$-element that is particularly attractive due to its simplicity and efficiency has been implemented in the ways *Nagtegaal* [2] and *Moran* [3] proposed. A geometrically nonlinear Q1-element and the geometrically linear Q1P0-element already available in SLang serve as reference. To examine the elements behavior during large deformations two-parameter Mooney-Rivlin and Neo-Hookian material laws are used.

When considering the specific properties of soft tissue one has to distinguish between passive tissue and mucle tissue with its activation dependant behavior. Based on *van Leeuven* [4] and *Johansson et.al.* [5] an anisotropic material model that allows for the consideration of recruitment schemes is explained.

[1] Pantuso, D., Bathe, K.-J. (1997). "On the stability of mixed finite elements in large strain analysis of incompressible solids". Finte Elements in Analysis and Design; 28:83-104

[2] Nagtegaal, J.C., Parks, D.M., Rice, J.R. (1974). "On numerically accurate finite element solutions in the fully plastic range". Computer Methods in Applied Mechanics and Engineering; 4:153-177

[3] Moran, B., Ortiz, M., Shih, C.F. (1990). "Formulation of implicit finite element methods for multiplicative finite deformation plasticity". International Journal for Numerical Methods in Engineering; 29:483-514

[4] Van Leeuven, J.L., Spoor, C.W. (1992). "Modelling mechanically stable muscle architectures". Philosophical Transactions of the Royal Society, London; 336:275-292

[5] Johansson, T., Meier, P., Blickhan, R. (2000). "A finite element model for the mechanical analysis of sceletal muscles". Journal of Theoretical Biology; 206:131-149

Multiscale Modeling of Woven Composites with Imperfect Microstructure

Jan ZEMAN, *Department of Structural Mechanics, Faculty of Civil Engineering, Czech Technical University in Prague, Czech Republic*

A comprehensive approach to the modelling of composite materials with a complex microstructure, introduced in [1–3], is discussed in the present contribution. Two different modelling strategies are examined. The first one assumes a well defined geometry of the microstructure arrangement and specific boundary conditions. In this framework, the complicated real microstructure is replaced by a substantially simpler periodic unit cell, which still statistically resembles the real microstructure. Periodic distribution of such unit cells is explored and the finite element method is called to carry out the numerical simulation. The theoretical basis for the second approach are the Hashin-Shtrikman variational principles. The random character of the fiber distribution is incorporated directly into the variational formulation employing certain statistical descriptors.

Several statistical descriptors suitable for the microstructure characterization are examined first. An appropriate optimization procedure is proposed to obtain the geometry of the desired unit cell. A number of numerical studies is performed to asses the quality of individual unit cells. It is demonstrated that the unit cell consisting of five fibers only provides reasonably accurate estimates of the macroscopic elastic properties. Similar conclusions apply to the thermal, linear and non-linear viscoelastic problems.

In certain applications the finite element approach used with the unit cell analysis may prove to be unnecessarily expensive. To overcome this limitation, the effective media theory based on variational principles of Hashin and Shtrikman, which incorporate the random character of fibers arrangement up the two-point probability, are used to predict the overall behavior of composites with statistically homogeneous microstructures. An extensive comparison with the finite element-based methodology is presented to investigate the applicability and limitations of this modelling strategy.

Finally, the extension of principles proposed for the determination of statistically optimized periodic unit cell to the modelling of plain-weave composites is proposed. The ability to incorporate a longitudinal shift of individual layers of the composite system into the model of the periodic unit cell is examined. Geometrical parameters derived from the optimization procedure are used to generate an equivalent periodic unit cell, which is then combined with the finite element method analysis to provide the effective material properties.

Acknowledgement: Financial support of this work was provided by the research project MSM:210000001

[1] Zeman, J. and Šejnoha, M., "Numerical evaluation of effective properties of graphite fiber tow impregnated by polymer matrix". *Journal of the Mechanics and Physics of Solids*, 49(1), 2001, pp. 69–90.

[2] Šejnoha, M. and Zeman, J., "Overall viscoelastic response of random fibrous composites with statistically quasi uniform distribution of reinforcements", *Computer Methods in Applied Mechanics and Engineering*, 191(44), 2002, pp. 5027–5044.

[3] Zeman, J. and Šejnoha, M., "Homogenization of plain weave composites with imperfect microstructure: Part I - Theoretical formulation", *International Journal of Solids and Structures*, 2003, submitted.

BRNO-WEIMAR-PRAGUE-WIEN (BOKU), 2nd PhD Workshop
Brno University of Technology
Faculty of Civil Engineering
Institute of Structural Mechanics
Veveří 95, 662 37 Brno, Czech Republic

GA ČR 103/03/1350
ISBN 80-214-2506-7
First Edition, November 2003