

# The Microplane Model for Concrete in COMSOL

A. Frigerio<sup>1</sup>

<sup>1</sup>RSE S.p.A.

\*via Rubattino 54 - 20134 Milan (Italy), [antonella.frigerio@rse-web.it](mailto:antonella.frigerio@rse-web.it)

**Abstract:** The safety of large civil structures is often evaluated by means of numerical models based on the Finite Element Method (FEM). In this frame, the choice of a constitutive law able to represent the complex mechanical behavior of concrete is a key point.

The classical approaches, whose theory is formulated in terms of stress and strain tensors and their invariants, are generally able to simulate in a proper way only a few specific characteristics of concrete, but they often present a limited capacity to reproduce the *overall* material behavior. A promising approach to overcome this limit seems to be the Microplane Model whose basic idea is related to the observation that the main mechanical phenomena of concrete take place on planes whose orientation depends on load and material conditions.

This paper deals with a detailed description of the implementation process followed to insert the elastic part of the Microplane Model within COMSOL. The examples used to validate this part are presented as well. The implementation of the non linear behavior of the microplane model is still in progress; therefore it is not addressed in this paper.

**Keywords:** Finite Element Method, concrete mechanical behavior, microplane model.

## 1. Introduction

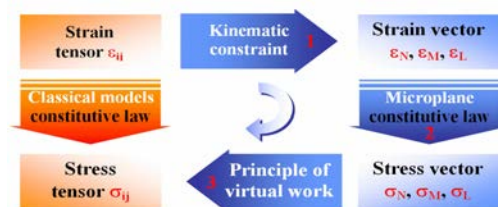
The theory the microplane model is based on is not of recent formulation as it came out from the pioneering idea of Taylor (1938), [1]. He proposed characterizing the constitutive behavior of polycrystalline metals by means of relations between the stress-strain vectors acting on planes of any orientation within the material and determining the macroscopic strain and stress tensors as a resultant of all these vectors, assuming static or kinematic constraint. The lack of computers first and their limited computational capacity afterwards prevented the practical application of this method in engineering problems up to few decades ago. In the meanwhile the microplane theory underwent some developments and it was adapted to deal with brittle materials such as concrete, [2]-[8].

In the next paragraphs the formulation of the microplane constitutive model for concrete will be briefly presented, followed by a detailed descriptions of its implementation within COMSOL. A simple example on a cube and a schematized concrete gravity dam will be used to show how the model works in COMSOL.

## 2. The microplane formulation for concrete

The philosophy governing the microplane model is essentially based on the experimental observation that the major mechanical phenomena characterizing concrete (creep, friction, shear and tensile fractures) are always referable to a plane whose orientation depends on material microstructure as well as loading and constraint conditions. Thus the microplane constitutive law is formulated by means of a relation between the strain and stress vectors acting on a plane of arbitrary orientation within the material. The orientation of this plane, called *microplane*, is defined by its unit normal  $\mathbf{n}$  of components  $n_i$ , where  $i$  refers to Cartesian components.

The microplane model follows a different logical scheme as regards classical approaches, anyway the starting point and the final one are obviously the same being respectively the strain and stress tensor, as shown in Figure 1. As a convention, in this paper the subscript N stands for the component of the strain vectors that are normal to microplanes, while M and L are referred to the tangential components.



**Figure 1.** The logical scheme of microplane model (in blu) compared with that of classical approaches (in orange).

Considering the scheme in Figure 1, three steps characterize the elastic behavior of the microplane model:

1. the strain vector on the microplane is the projection of the macroscopic strain tensor (kinematic constraint);
2. the stress vector is computed by means of the microplane constitutive law;
3. the macroscopic stress tensor is computed applying the principle of virtual work.

In the following paragraphs each step will be briefly described.

The non linear behavior of the microplane model is not addressed in this paper because its implementation within COMSOL is still in progress.

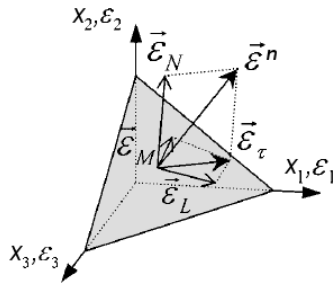
### 2.1 First Step: application of the kinematic constraint

At first the number and orientation of all microplanes related to a generic material point should be defined. In literature different solutions are provided and their accuracy is specified as well, [3].

Considering each material point, the kinematic constraint is applied projecting the strain tensor  $\varepsilon_{ij}$  on each microplane (Figure 2):

$$\varepsilon_i^k = \varepsilon_{ij} n_j \quad (1)$$

being  $n_j$  the unit vector components normal to the microplane  $k$ , [6].



**Figure 2.** Components of the strain vector  $\varepsilon^n$  referred to a generic microplane:  $\varepsilon_N$ ,  $\varepsilon_L$ ,  $\varepsilon_M$ .

The component of the strain vector normal to the microplane is given by:

$$\varepsilon_N = n_i \varepsilon_i^k = n_i (\varepsilon_{ij} n_j) = N_{ij} \varepsilon_{ij} \quad (2)$$

being the tensor  $N_{ij} = n_i n_j$ .

In order to compute the shear strain components, two orthogonal directions should be determined by means of two orthogonal unit vectors  $m$  and  $l$ , both lying on the same microplane:

$$\varepsilon_M = m_i (\varepsilon_{ij} n_j) = M_{ij} \varepsilon_{ij} \quad (3)$$

$$\varepsilon_L = l_i (\varepsilon_{ij} n_j) = L_{ij} \varepsilon_{ij} \quad (4)$$

where  $M_{ij}$  and  $L_{ij}$  are symmetric tensors, defined as follows:

$$M_{ij} = \frac{m_i n_j + m_j n_i}{2} \quad (5)$$

$$L_{ij} = \frac{l_i n_j + l_j n_i}{2} \quad (6)$$

### 2.2 Second Step: the microplane constitutive law

The constitutive law that allows computing the microplane stresses from the microplane strains usually splits the normal strain vectors into their volumetric and deviatoric parts.

$$\sigma_N = \sigma_V + \sigma_D; \quad \varepsilon_N = \varepsilon_V + \varepsilon_D \quad (7a,b)$$

where the volumetric stress  $\sigma_V$  as well as the corresponding volumetric strain  $\varepsilon_V$  are equal for all microplanes:

$$\sigma_V = \frac{\sigma_{kk}}{3}; \quad \varepsilon_V = \frac{\varepsilon_{kk}}{3} \quad (8a,b)$$

On the microplane level, the elastic response of the constitutive law is defined by means of the elastic incremental relations in the rate form:

$$\dot{\sigma}_V = E_V \dot{\varepsilon}_V; \quad \dot{\sigma}_D = E_D \dot{\varepsilon}_D \quad (9a,b)$$

$$\dot{\sigma}_M = E_M \dot{\varepsilon}_M; \quad \dot{\sigma}_L = E_L \dot{\varepsilon}_L \quad (10a,b)$$

where  $E_V$ ,  $E_D$  and  $E_T$  are the microplane elastic moduli:

$$E_V = \frac{E}{1-2\nu}; \quad E_D = \frac{5E}{(2+3\mu)(1+\nu)} \quad (11a,b)$$

$$E_T = \mu E_D \quad (12)$$

being  $E$  the Young modulus,  $\nu$  the Poisson ratio and  $\mu$  a parameter of the method, generally assumed unitary.

### 2.3 Third Step: application of the principle of virtual work

The stress components on each microplane are not in general equal to the projections of

macroscopic stress tensor  $\sigma_{ij}$  if the kinematic constraint has been applied to the strain tensors  $\epsilon_{ij}$ . Because of that, the static equilibrium is imposed by means of the principle of virtual work written with reference to the surface  $\Omega$  of a unit hemisphere whose center is supposed to be the material point. Considering that the variation equation of this principle must hold for any variation  $\delta\epsilon_{ij}$ , the following basic equilibrium equation is attained:

$$\sigma_{ij} = \frac{3}{2\pi} \int_{\Omega} (\sigma_N \cdot N_{ij} + \sigma_M \cdot M_{ij} + \sigma_L \cdot L_{ij}) d\Omega \quad (13)$$

The integral over the surface of the unit hemisphere represents the integration over an infinite number of microplanes but its solution is not a trivial matter. For this reason, Gaussian quadrature formulas of various degrees of approximation are adopted:

$$\sigma_{ij} \approx 6 \sum_{k=1}^{N_{mp}} w_k (\sigma_N \cdot N_{ij} + \sigma_M \cdot M_{ij} + \sigma_L \cdot L_{ij})^{(k)} \quad (14)$$

This approximation represents a weighted summation over all  $N_{mp}$  microplanes in which normalized weights  $w_k$  are adopted.

In literature, it was assessed that a good arrangement between an accurate solution of the integration formula and the computational cost of the overall method in terms of time and numerical resources is assured when 28 microplanes are defined for each material point, [3].

### 3. Implementation of the microplane model in COMSOL

In most software the implementation of the microplane model implies the development of complex external user subroutines. The setting of the interactions occurring between software and subroutines is very often a challenging task.

On the contrary, COMSOL does not necessarily require developing subroutines since it provides a user-friendly platform. For this reason, together with its capacity to deal with multi-physic problems, COMSOL was deemed a good platform to implement the microplane model.

The implementation process was arranged in several phases, defining respectively:

- the global parameters and variables that are common to all microplanes;

- the strain and stress vectors according to the theoretic assumptions;
- the constitutive law governing mechanical quantities at the microplane level.

In order to speed up the implementation process, the variables as well as the vector and tensor quantities were defined within the environment of MATLAB code which is well interfaced with COMSOL. The *comsol server matlab* command was used.

#### 3.1 Global parameters and variables

Many parameters and variables of the method are common to all microplanes; thus, it is useful to compute them all just once before starting any analyses.

Making reference to the *Model Builder* window of COMSOL, all these quantities were placed within the *Global Definitions* node (Figure 3); in particular, they are:

- the material and model parameters (i.e. Young modulus, Poisson's ratio, concrete density and the microplane elastic moduli as defined in §2.2);
- the direction cosines  $n$ ,  $m$  and  $l$  of all microplanes (§2.1);
- the weights  $w_k$  to be applied to the 28 microplanes in the adopted Gaussian integration formula (§2.3);
- the tensor  $N_{ij}$ ,  $M_{ij}$  and  $L_{ij}$  used to project the strain tensor  $\epsilon_{ij}$  on the 28 microplanes of each material point (§2.1);
- the three unit versors of Cartesian axes.

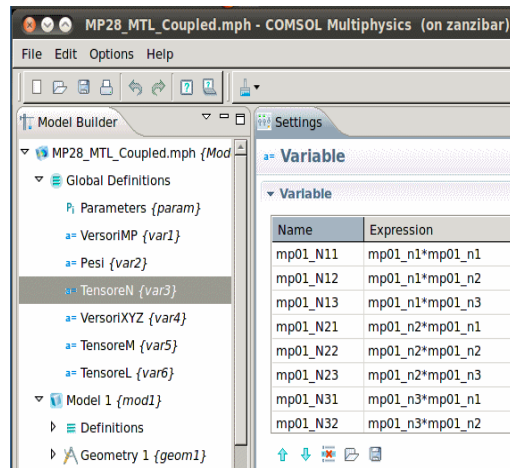


Figure 3. Example relevant to the definition of some variables within the *Global Definition* node.

The direction cosines  $n$ ,  $m$  and  $l$  as well as the weights  $w_k$  related to the 28 microplanes are available in literature [3].

In order to assign meaningful names to all these variables, common conventions were considered: the first letters of each name identify the microplane the variable is referred to ( $mp00$ , with  $00 = 1 \div 28$ ), while the second part of the name specifies the type of variable as well as the Cartesian axes it is correlated to. Therefore, the generic name of each variable was defined as follows:

- $mp00\_nj$ ,  $mp00\_mj$ ,  $mp00\_lj$  stand for the direction cosines  $n$ ,  $m$  and  $l$  related to the Cartesian axis  $j$ ;
- $mp00\_w$  represents the weights of the Gaussian formula;
- $mp00\_Nij$ ,  $mp00\_Mij$ ,  $mp00\_Lij$  denote the tensor  $N_{ij}$ ,  $M_{ij}$  and  $L_{ij}$ .

### 3.2 Strain and stress vectors

Either the strain or stress vectors were specified at a local level in the *Definitions* node that is part of the *Model* node in which the structure to be studied has to be fully described and characterized.

Considering that the constitutive law is expressed in the rate form, the time derivatives of the strain vectors were defined. In the *eNML* node the expression of the three components of the strain vector of each microplane were included (Figure 4). The name of every strain component was linked to the corresponding microplane as follows:  $eN00$ ,  $eM00$ ,  $eL00$ . The normal component was also split in its volumetric and deviatoric part:  $eV00$ ,  $eD00$ .

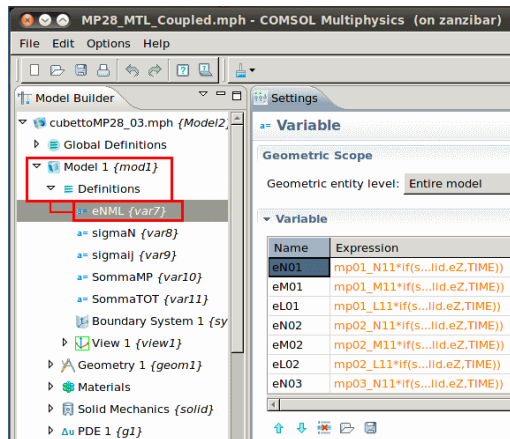


Figure 4. Definition of the time derivatives of the three components of the strain vectors.

The outputs given by the microplane constitutive law were expressed in terms of volumetric, deviatoric and shear stress components. Starting from these vector components the corresponding stress tensor was attained applying the principle of virtual work. Considering equation (14), within the  $\sigma_{mij}$  node the summation of the stress tensor components was computed for each microplane (Figure 5). In this case a different notation was assumed to identify these quantities: the first letters of the name indicate the stress tensor component while the second part the microplane it is referred to:  $sij\_00$ .

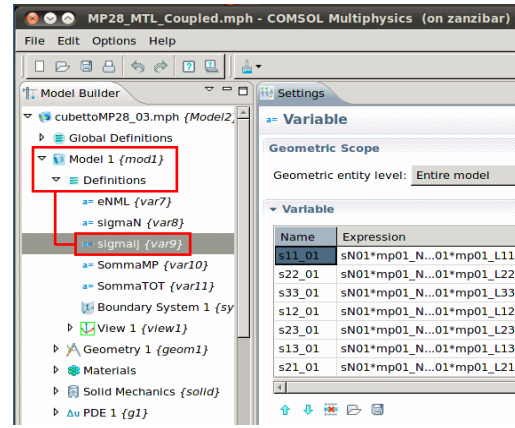


Figure 5. Definition of stress tensor components of each microplane.

Within the *SommaMP* node the normalized weights were applied, while in the *SommaTOT* node the resulting stress tensor components for material points were defined (Figure 6).

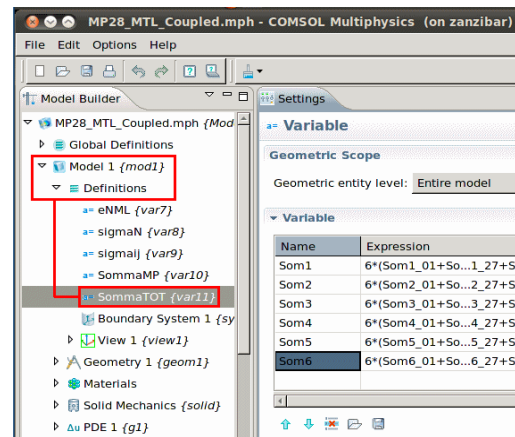


Figure 6. Definition of the resulting stress tensor components for a generic material point.

Since a symmetric tensor is considered, the six components were respectively named:

- Som1 =  $\sigma_{xx}$ ;
- Som2 =  $\sigma_{yy}$ ;
- Som3 =  $\sigma_{zz}$ ;
- Som4 =  $\sigma_{xy}$ ;
- Som5 =  $\sigma_{xz}$ ;
- Som6 =  $\sigma_{yz}$ ;

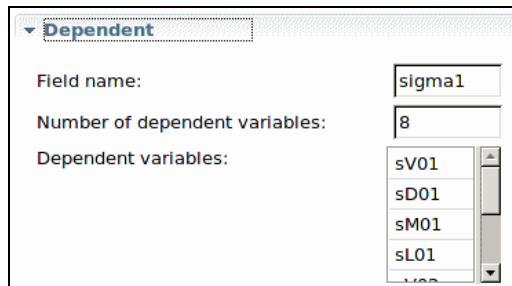
### 3.3 Constitutive law at the microplane level

The constitutive law was implemented in COMSOL by means of its PDE modules. More than one PDE module was used just to manage in a better way all equations that have to be defined for each material point, that is a total of 112 equations resulting from the four vector components per 28 microplanes.

In the *Setting* window of each PDE module the *General Form PDE* interface allows specifying the coefficients for a general PDE form. In this case, the PDE coefficients were set up according to the microplane constitutive law expressed in terms of elastic relations in the rate form as specified in equations (9a,b) and (10a,b). The default equation provided by COMSOL is as follows:

$$e_a \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot \Gamma = f \quad (15)$$

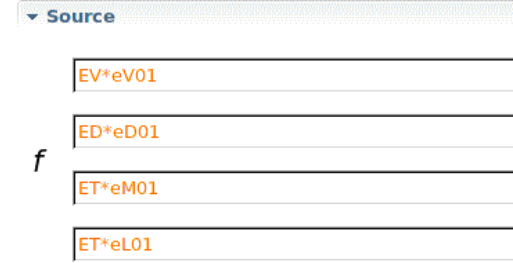
In the present work, the stress vector components were assumed as the dependent variables  $u$ . In analogy to the corresponding strain vector components they were named:  $sV00$ ,  $sD00$ ,  $sM00$  and  $sL00$  (Figure 8).



**Figure 8.** Definition of dependent variables in the *Setting* window of the PDE module.

The second time derivatives as well as the flux vector  $\Gamma$  are not present in the microplane constitutive law; therefore, all terms of matrix  $e_a$  and of flux vector  $\Gamma$  were assumed equal to zero. Considering that in the elastic relations of each microplane the dependent variables are

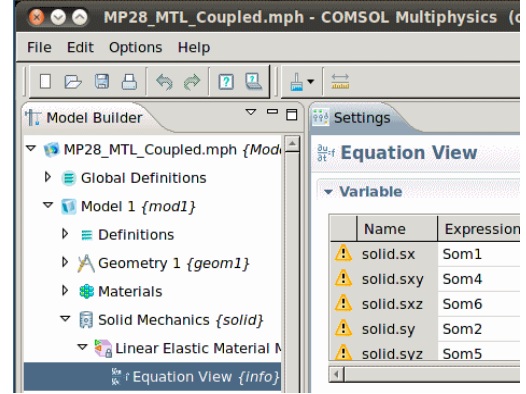
involved one at a time, matrix  $d_a$  is a unit matrix, while the source terms  $f$  are equal to the right hand terms of equations (9a,b) and (10a,b) as shown in Figure 9.



**Figure 9.** Definition of the source terms of the PDE form in the *Setting* window of the PDE module.

The *Solid Mechanics* module was added as well because the microplane model is used to study the mechanical behavior of concrete structures. In this case, the dependent variables are the displacement field:  $u$ ,  $v$  and  $w$ .

In order to couple the *Solid Mechanics* module with the PDE modules, the values relevant to the stress tensor components as computed in the PDE modules were assigned to the corresponding components of the mechanical stress tensor (Figure 10).



**Figure 10.** Coupling between the *Solid Mechanics* module and the PDE modules.

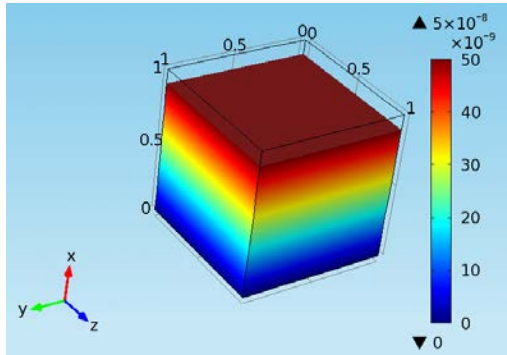
## 4. Simple applications

So far only the elastic behavior of the microplane model has been implemented and verified, while the implementation of the non linear behavior is still in progress.

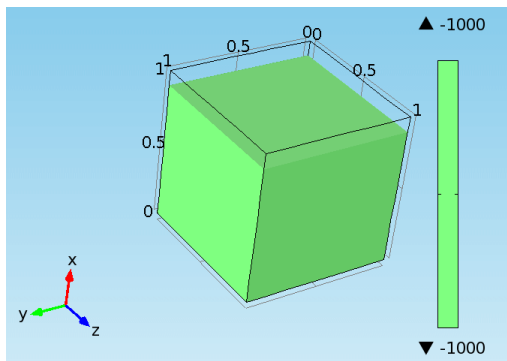
The microplane model was applied on a concrete cube with one meter long edge, loaded on one face by means of a 1000 N/m<sup>2</sup> constant pressure and constrained on the opposite side. The resulting displacement along the loaded direction is 5.E-08 m (Figure



11) while the corresponding stress component is equal to  $1000 \text{ N/m}^2$  (Figure 12). Both values can be manually attained taking into account a Young modulus of  $2.E+10 \text{ N/m}^2$ .



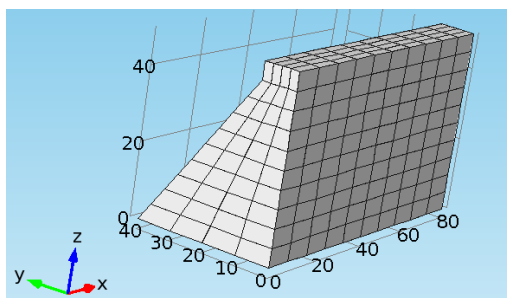
**Figure 11.** Displacement field along the loaded direction.



**Figure 12.** Contour of the stress tensor component along the y direction (i.e. Som2).

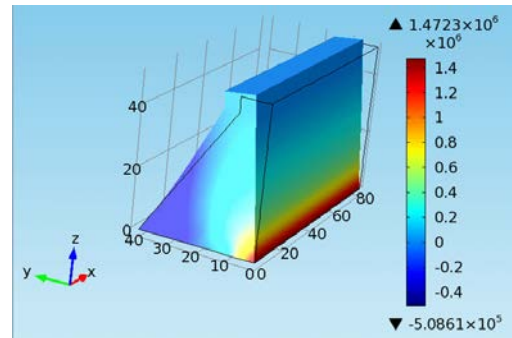
This simple example was useful to evaluate the correctness of the implementation process in COMSOL. Either the *Fully Coupled* time-dependent solver or the *Segregated* one gives the exact solution.

An engineering application was studied as well taking into consideration a schematized geometry of a concrete gravity dam. The mesh used to study this case is reported in Figure 13.

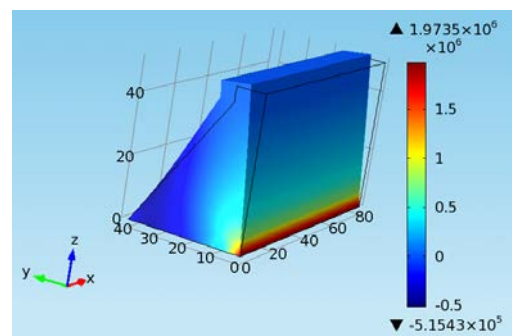


**Figure 13.** Mesh of a concrete gravity dam.

The field of vertical stresses is qualitatively in good agreement with that computed using a simple linear elastic model, anyway a slight difference could be observed in the extreme values of the contour (Figure 14 and 15). This difference is still under investigation in order to understand if it is due to the application of the kinematic constraint as well as the principle of virtual work. The tolerance used to solve the problem is also under control.



**Figure 14.** Contour of the vertical stress tensor component (i.e. Som3) in case a microplane model is used.



**Figure 15.** Contour of the vertical stress tensor component in case a linear elastic model is used.

## 5. Conclusions

The ability to properly simulate the overall behavior of concrete in order to assess the safety of large civil structures is actually important. In this context, the microplane model was proved to be a good solution to represent the complex behavior of concrete; moreover, the advancements in computer performances have allowed the application to study also engineering problems.

The elastic behavior of the microplane model was implemented within COMSOL and verified by means of simple examples. The implementation process was described in details making reference to the theory the method is based on.

A good setting of this elastic phase was crucial to work in the non linear behavior.

## 6. References

1. Taylor, G.I., Plastic strain in metal, *J. Inst. of Metals*, **62**, 307-324 (1938)
2. Bažant, Z.P., Microplane model for strain controlled inelastic behaviour, Chapter 3. *C.S. Desai and R.H. Gallagher eds.*, Wiley, London (1984)
3. Bažant, Z.P. and Oh, B., Microplane model for progressive fracture of concrete and rock, *ASCE Journal of Engineering Mechanics*, **111(4)**, 559-582 (1985)
4. Bažant, Z.P. and Prat, P., Microplane model for brittle-plastic material: I. Theory, *Journal of Engineering Mechanics*, **114(10)**, 1672-1688 (1988)
5. Bažant, Z.P. and Prat, P., Microplane model for brittle-plastic material: II. Verification, *Journal of Engineering Mechanics*, **114(10)**, 1689-1702 (1988)
6. Bažant, Z.P., Xiang, Y. and Prat, P., Microplane model for concrete: I. Stress-strain boundaries and finite strain, *Journal of Engineering Mechanics*, **122(3)**, 245-262 (1996)
7. Bažant, Z.P., Xiang, Y., Adley, M. and Prat, P., Microplane model for concrete: II. Data delocalization and verification, *Journal of Engineering Mechanics*, **122(3)**, 263-268 (1996)
8. Bažant, Z.P. and Carol, I., Damage and plasticity in microplane theory, *International Journal of Solids and Structures*, **34(29)**, 3807-3835 (1997)

## 7. Acknowledgements

This work has been financed by the Research Fund for the Italian Electrical System under the Contract Agreement between RSE and the Ministry of Economic Development.