

Stuttgart (Germany) 26<sup>th</sup>-28<sup>th</sup> October 2011

# The Microplane Model for Concrete in COMSOL

*A. Frigerio*

---

# Agenda

---

- Aims of the work
- The Microplane Model
  - Main theory aspects
- Implementation in COMSOL
  - How the elastic behaviour was developed
- Few simple examples
- Conclusions

# Why a new model for concrete?

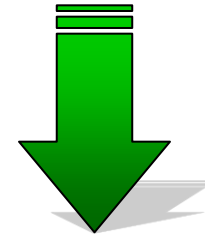
---

- Safety of large civil structures:
  - Often evaluated by means of numerical models based on the Finite Element Method (FEM)
    - Classical approaches are able to properly simulate *only a few specific* characteristic of concrete
    - The Microplane Model seems to be a promising alternative to represent the *overall* behaviour of concrete

# Microplane model: theory

---

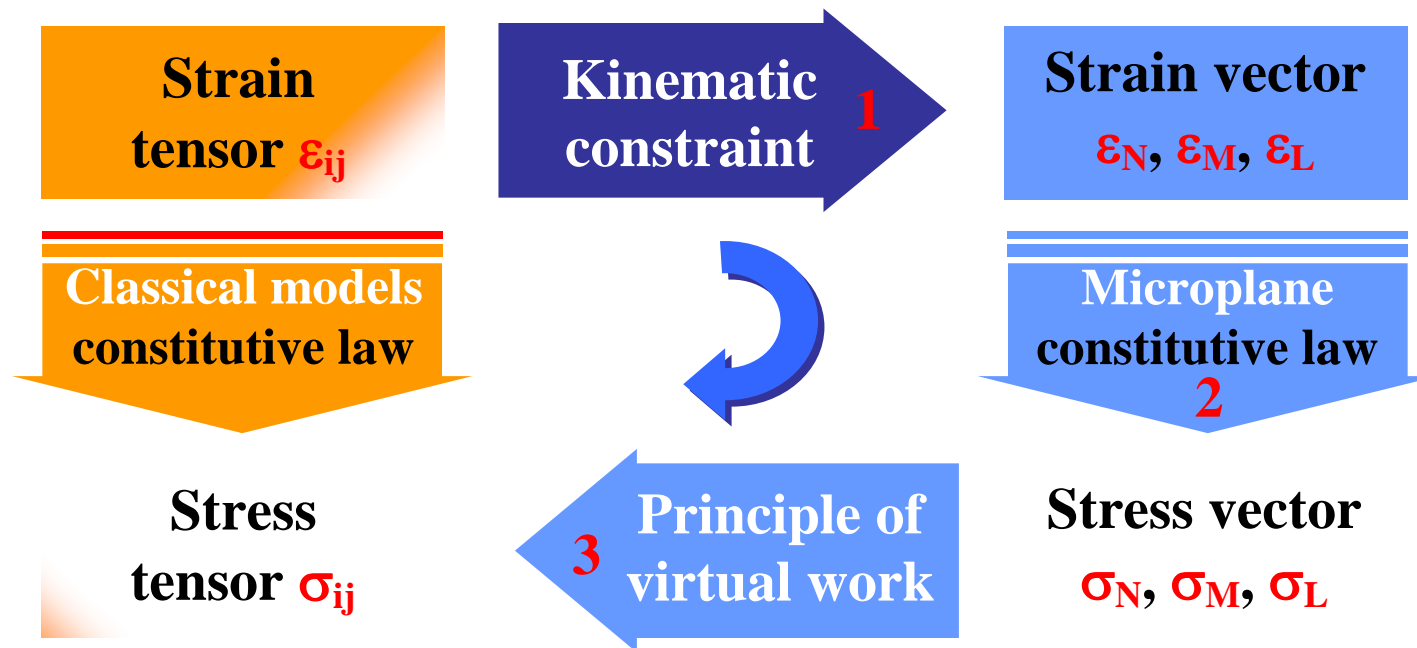
- The major mechanical phenomena of concrete are always referable to a plane whose orientation depends on material microstructures, as well as loading and constraint conditions



- The microplane constitutive law is formulated by means of a relation between the **strain and stress vectors** acting on a **plane whose orientation is arbitrary**

# Microplane model: theory

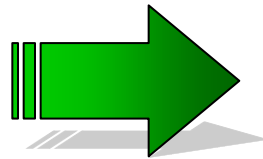
- The microplane logical scheme consists of **three parts**
  - The starting point and the final one are the same of classical approaches



# Microplane model: theory

- **Step 1: application of the kinematic constraint**
  - The number and orientation of all microplanes related to a material point should be defined
  - In each material point the strain tensor  $\varepsilon_{ij}$  is projected on each microplane

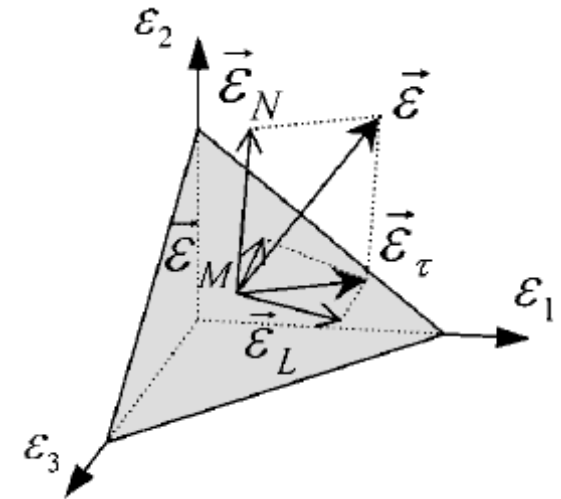
$$\varepsilon_i^k = \varepsilon_{ij} n_j$$



$$\varepsilon_N = n_i (\varepsilon_{ij} n_j) = N_{ij} \varepsilon_{ij}$$

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

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



# Microplane model: theory

- **Step 2: the microplane constitutive law**
  - The normal strain and stress vectors are split into their volumetric and deviatoric parts
  - The volumetric strain and stress are equal for all microplanes
  - The elastic response is defined by means of the elastic incremental relations in the rate form:

$$\dot{\sigma}_V = E_V \dot{\epsilon}_V \quad \dot{\sigma}_D = E_D \dot{\epsilon}_D \quad \dot{\sigma}_M = E_M \dot{\epsilon}_M \quad \dot{\sigma}_L = E_L \dot{\epsilon}_L$$

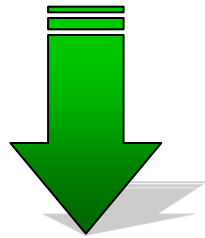
$$E_V = \frac{E}{1-2\nu} \quad E_D = \frac{5E}{(2+3\mu)(1+\nu)} \quad E_T = \mu E_D$$

# Microplane model: theory

- **Step 3: application of the principle of virtual work**

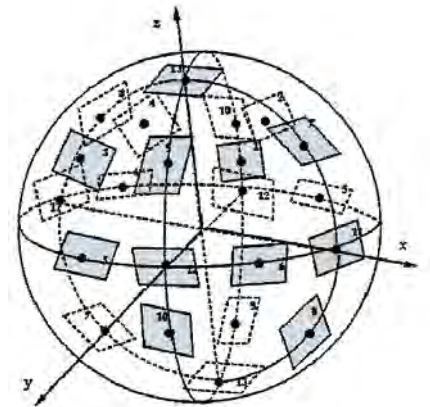
- The static equilibrium is written with reference to the surface  $\Omega$  of a unit hemisphere whose centre is the material point

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



Gaussian quadrature formulas of various degrees of approximation are used to solve the integral over the unit hemisphere

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





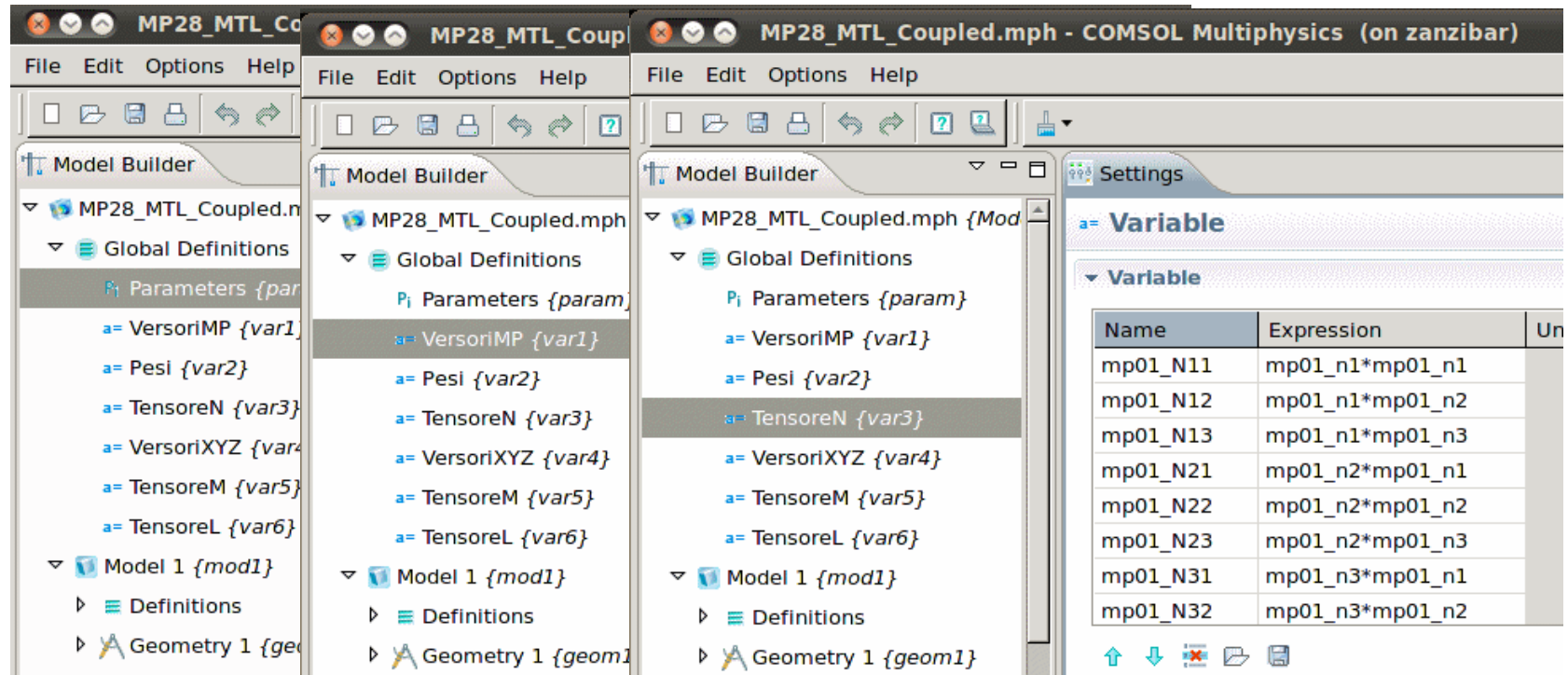
# Implementation in COMSOL

---

- COMSOL does not necessarily require developing subroutines thanks to its user-friendly platform
- The implementation process was arranged in several phases, defining:
  - The global parameters and variables that are referred to all microplanes and they are common to all material points
  - The strain and stress vectors on each microplane
  - The constitutive law governing mechanical quantities at the microplane level

# Implementation in COMSOL

- Global parameters and variables definition
  - In the **Global Definition node** of the Model Builder window

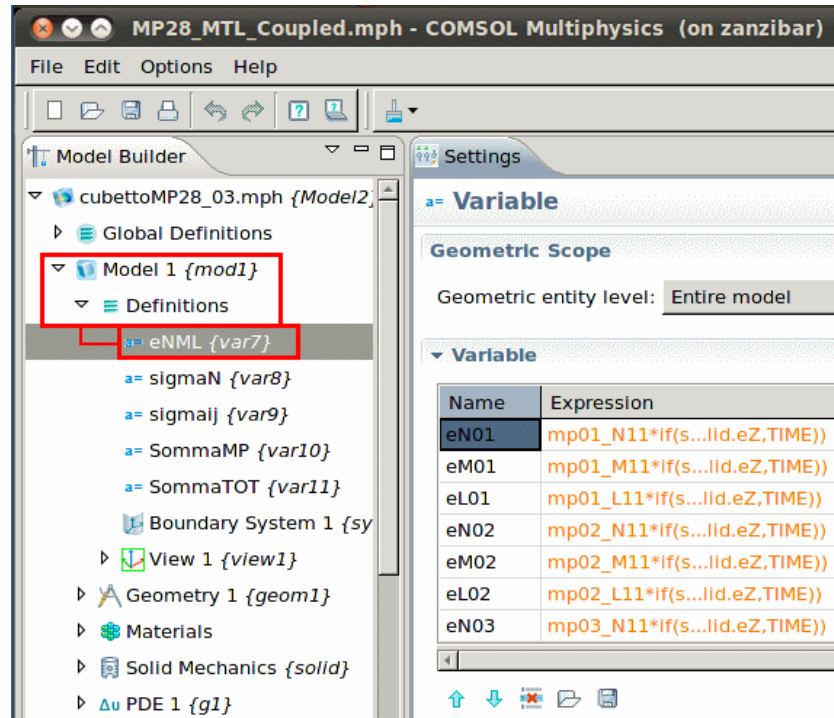


The image displays three screenshots of the COMSOL Model Builder window, illustrating the implementation of global parameters and variables. The first two screenshots show the 'Global Definitions' node expanded, revealing a list of parameters and variables. The third screenshot shows the 'Settings' dialog for a variable, displaying a table of variable names and their expressions.

Name	Expression	Un
mp01_N11	mp01_n1*mp01_n1	
mp01_N12	mp01_n1*mp01_n2	
mp01_N13	mp01_n1*mp01_n3	
mp01_N21	mp01_n2*mp01_n1	
mp01_N22	mp01_n2*mp01_n2	
mp01_N23	mp01_n2*mp01_n3	
mp01_N31	mp01_n3*mp01_n1	
mp01_N32	mp01_n3*mp01_n2	

# Implementation in COMSOL

- Strain vectors definition
  - In the **Definition node** that is a part of the **Model node**



$$\varepsilon_N = n_i (\varepsilon_{ij} n_j) = N_{ij} \varepsilon_{ij}$$

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

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

- All strain vector components are defined in the rate form

# Implementation in COMSOL

- Constitutive law implemented by means of **PDE modules**
  - A total of 112 equations resulting from the 4 vector components per 28 microplanes in each material point
  - In the **General Form PDE** of the **Setting window**:

$$\cancel{e_a \frac{\partial^2 \mathbf{u}}{\partial t^2}} + d_a \frac{\partial \mathbf{u}}{\partial t} + \cancel{\nabla \cdot \Gamma} = f$$



$$\dot{\sigma}_{V,D,M,L} = E_{V,D,T,T} \dot{\epsilon}_{V,D,M,L}$$

The screenshot shows the 'General Form PDE' settings window. The 'Dependent' section is expanded, showing 'Field name: sigma1' and 'Number of dependent variables: 8'. Below this, a list of dependent variables includes sV01, sD01, sM01, and sL01. The 'Source' section is also expanded, showing the expression 'EV\*eV01 + ED\*eD01 + ET\*eM01 + ET\*eL01' for the variable 'f'.

# Implementation in COMSOL

- Stress vectors definition
  - In the **Definition node** that is a part of the **Model node**

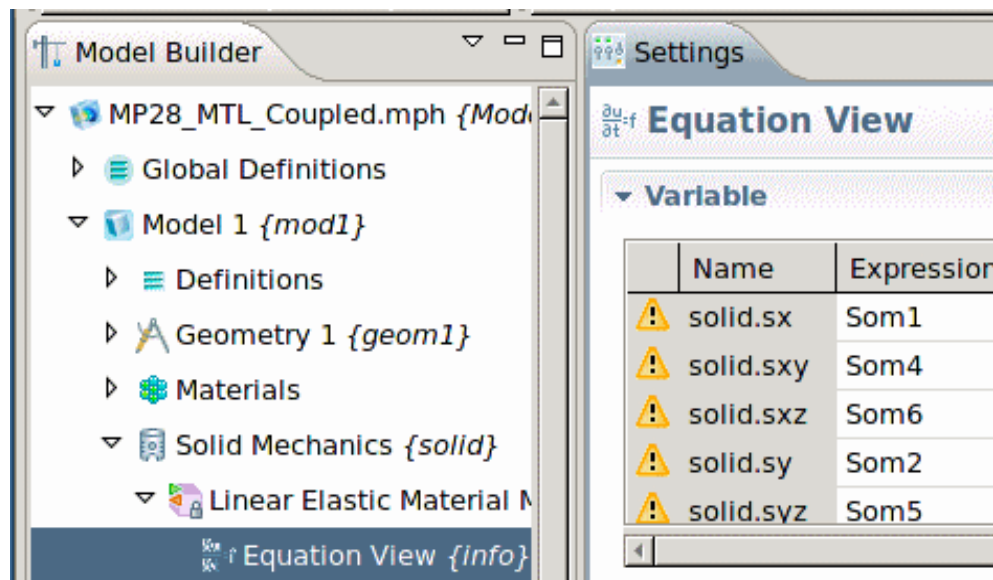
$$\sigma_{ij} \approx 6 \sum_{k=1}^{N_{mp}} w_k \left( \sigma_N \cdot N_{ij} + \sigma_M \cdot M_{ij} + \sigma_L \cdot L_{ij} \right)^{(k)}$$

The image shows two screenshots of the COMSOL Multiphysics interface, illustrating the implementation of the stress vector definition. The left screenshot shows the 'Model 1 {mod1}' node expanded to 'Definitions', with 'sigmaIj {var9}' selected. The right screenshot shows the 'SommaTOT {var11}' node selected. Both screenshots show the 'Variable' table with the expression for the stress vector.

Name	Expression
s11_01	sN01*mp01_N...01*m
s22_01	sN01*mp01_N...01*m
s33_01	sN01*mp01_N...01*m
s12_01	sN01*mp01_N...01*m
s23_01	sN01*mp01_N...01*m

# Implementation in COMSOL

- The **Solid Mechanics module** was added to study the mechanical behaviour of concrete structures
  - The dependent variables are the displacement field:  $u$ ,  $v$  and  $w$
  - This module is coupled with the PDE modules as follows:

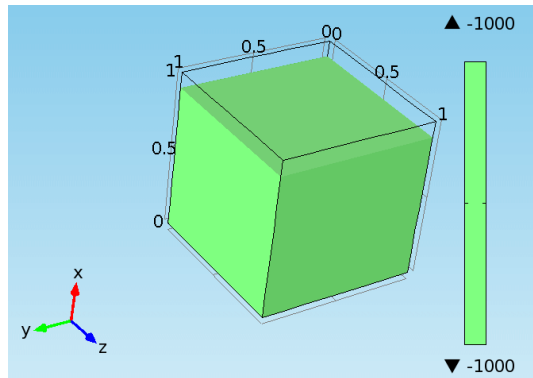


- Som1 =  $\sigma_{xx}$
- Som2 =  $\sigma_{yy}$
- Som3 =  $\sigma_{zz}$
- Som4 =  $\sigma_{xy}$
- Som5 =  $\sigma_{xz}$
- Som6 =  $\sigma_{yz}$

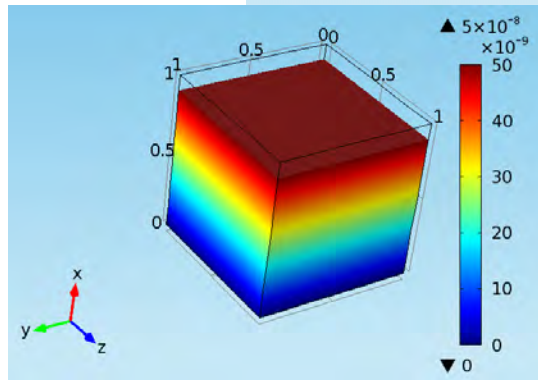
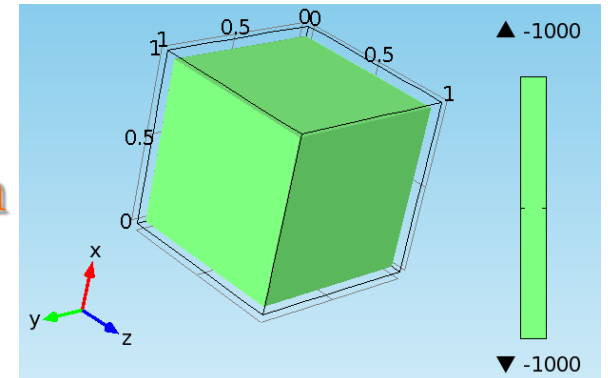
# Simple applications

- The elastic behaviour was verified on a concrete cube

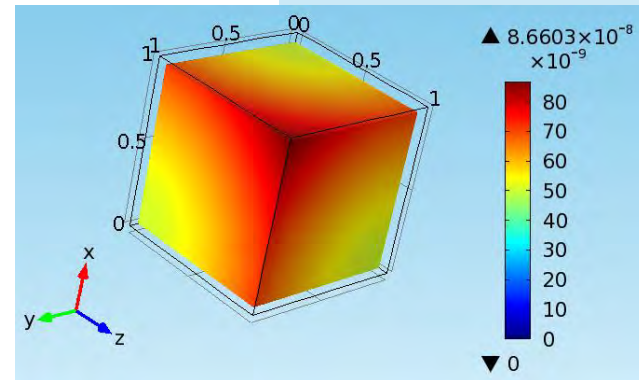
Uniaxial  
compression



Triaxial  
compression



$$E = 2.0e+10 \text{ N/m}^2$$
$$p = 1000 \text{ N/m}^2$$

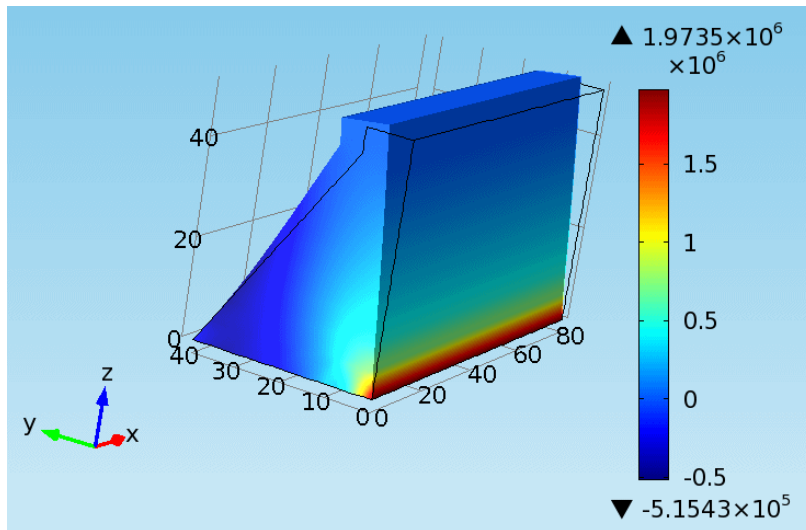




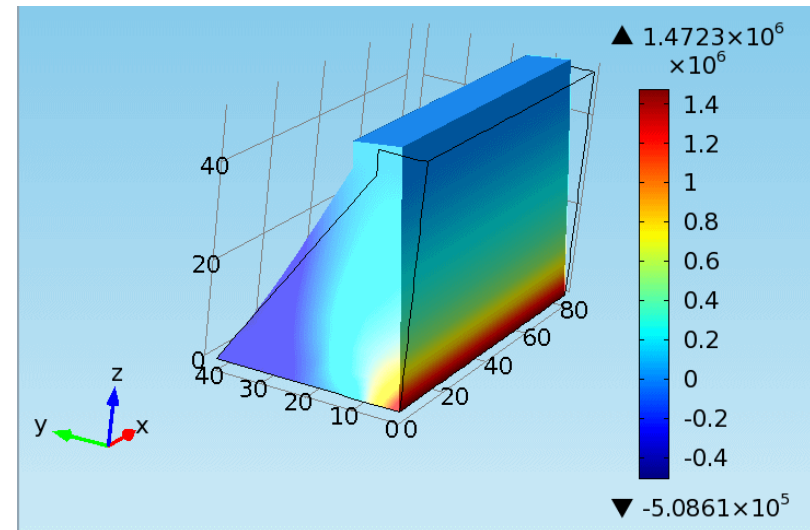
# Simple applications

- The elastic behaviour was verified on a concrete gravity dam

## Linear elastic model



## Microplane model





# Conclusions

---

- COMSOL has shown a good capacity to manage a complex implementation process
- 3D constitutive laws could be easily implemented without writing complex user subroutines
- The implementation of the non-linear behaviour of concrete is in progress
  - No external user subroutines are required
  - The implementation methodology is similar to that used for the linear elastic part

# *Thank you*

[antonella.frigerio@rse-web.it](mailto:antonella.frigerio@rse-web.it)

*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.*