Using External Material Functions in the Structural Mechanics Module

Mazars' Damage Model

COMSOL

Overview

- A new way to specify user-defined material models is included in COMSOL Multiphysics version 5.2.
- You can now access external material functions, written in C code, which have been compiled into a shared library.
- This makes it possible to program your own material models and distribute such models as add-ons.
- Examples include a model file, a source code file, and a shared library compiled and linked for 64-bit Windows
- Running the models on Linux™ and OS X requires additional compilation and linking

Mazars' Damage Model

- In the continuum damage mechanics formalism, a "damage" variable represents the amount of deterioration due to crack growth. Damage plays an essential role in the weakening of the material's stiffness.
- Mazars' model describes the fracture behavior in concrete by an isotropic scalar damage variable, which enters the constitutive stress-strain relationship:

$$\sigma = (1 - d)\, C : \epsilon$$

- The damage is described by two different evolution laws for the development of damage under tension and compression.
- The total damage is given by a linear combination of these two:

$$d = \alpha_t * d_t + \alpha_c * d_c$$

Mazars' Damage Model

- Tensile damage is given by the relation:

$$d_t(\kappa) = 1 - \frac{\kappa_0(1 - A_t)}{\kappa} - A_t e^{-B_t(\kappa - \kappa_0)}$$

- Compressive damage is given by the relation:

$$d_c(\kappa) = 1 - \frac{\kappa_0(1 - A_c)}{\kappa} - A_c e^{-B_c(\kappa - \kappa_0)}$$

- $A_t, B_t, \kappa_0, A_c, B_c$ are material parameters.
- κ is a state variable to memorize the maximum value of the effective tensile strain, which is defined by the principal strains in tension

$$\epsilon_{ef} = \sqrt{\langle \epsilon_1 \rangle^2 + \langle \epsilon_2 \rangle^2 + \langle \epsilon_3 \rangle^2}$$

- Here $\langle \cdot \rangle$ means the Macaulay brackets (negative values replaced by zero).
- κ is updated (and therefore the damage increases) only if ϵ_{ef} exceeds its historical maximum. The state κ is initialized to κ_0.

COMSOL

Mazars' Damage Model

- The tensile and compressive weights, α_t and α_c, are derived from principal stresses
- The tensile principal stresses and the material stiffness define the tensile principal strains

$$\epsilon_t = C^{-1}\langle\sigma\rangle$$

- These define the weights from

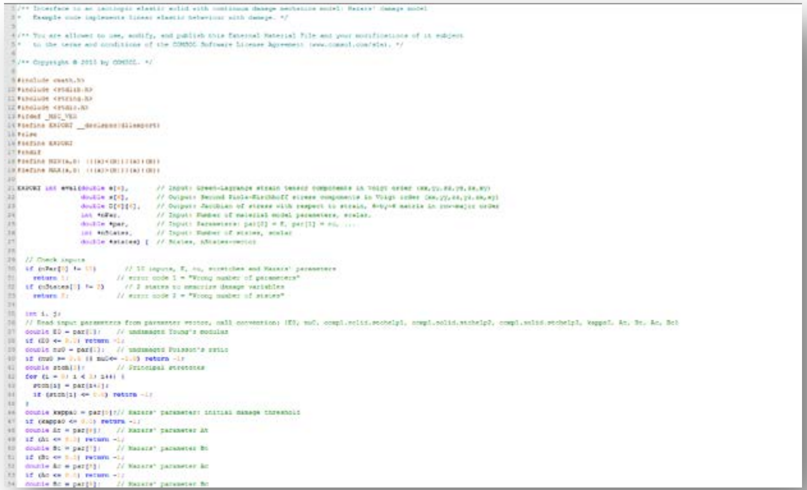
$$\alpha_t = \sum_{i=1}^{3} H_i \epsilon_{ti} \epsilon_i / {\epsilon_{ef}}^2$$

$$\alpha_c = \sum_{i=1}^{3} H_i \epsilon_{ci} \epsilon_i / {\epsilon_{ef}}^2$$

- The compressive strains are computed from the difference between principal strains and tensile strains: $\epsilon_{ci} = \epsilon_i - \epsilon_{ti}$
- H_i depend on the principal strains: $H_i = 1$ if $\epsilon_i > 0, 0$ otherwise.

Mazars' Damage Model

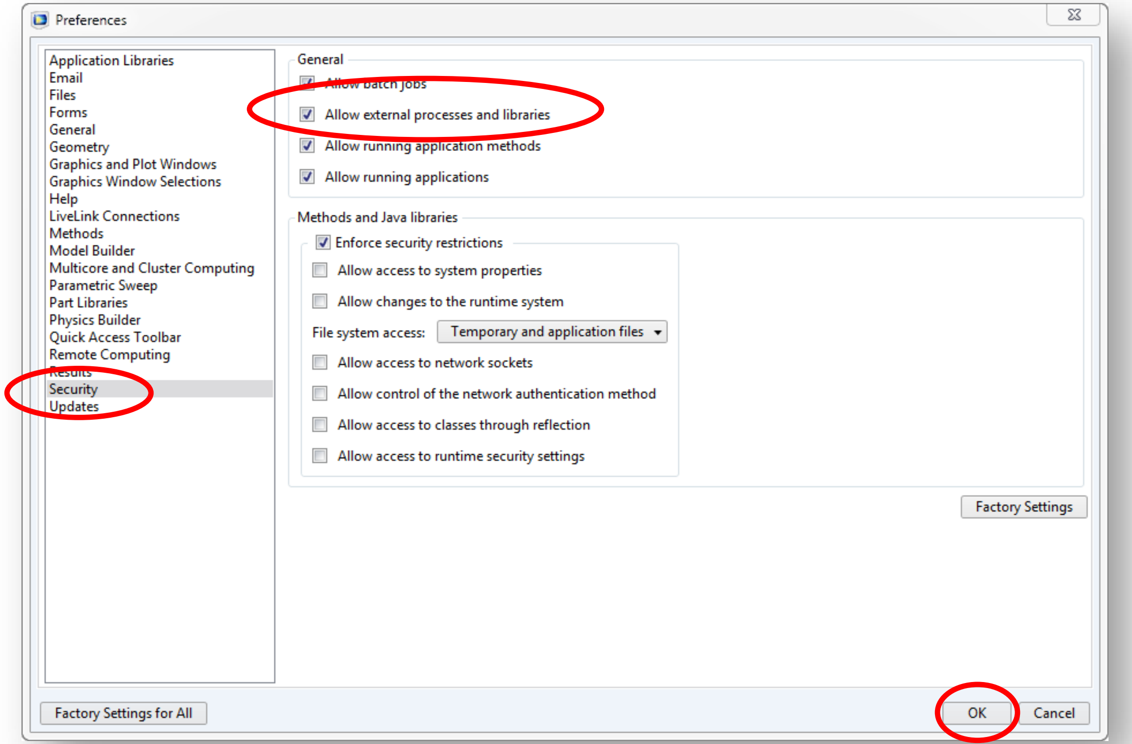
- Mazars' damage model is implemented as a C-function compiled and linked to create dynamically linked libraries that can be called from a material node in the Model Builder at runtime.



- For details, see the section *Working with External Materials* in the *COMSOL Multiphysics Reference Manual*.

Allow External Processes and Libraries

- For security reasons, executing external code is by default not allowed in a new COMSOL installation
- Open the **Preferences** dialog box, go to **Security** and select **Allow external processes and libraries**
- Restart COMSOL Multiphysics



Example

- Compute the axial stress for uniaxial tension and compression for a solid under Mazars' damage model and compare the results to published data.
- Use compiled library Mazars_damage.dll.
- Add a Solid Mechanics interfaces on a simple 2D axisymmetric geometry.
- Uniaxial tension and uniaxial compression computed in two Study Steps.
- Material parameters from "Mechanical Behavior of Concrete", Torrenti, Pijaudier-Cabot and Reynouard eds. Wiley 2010.

Settings

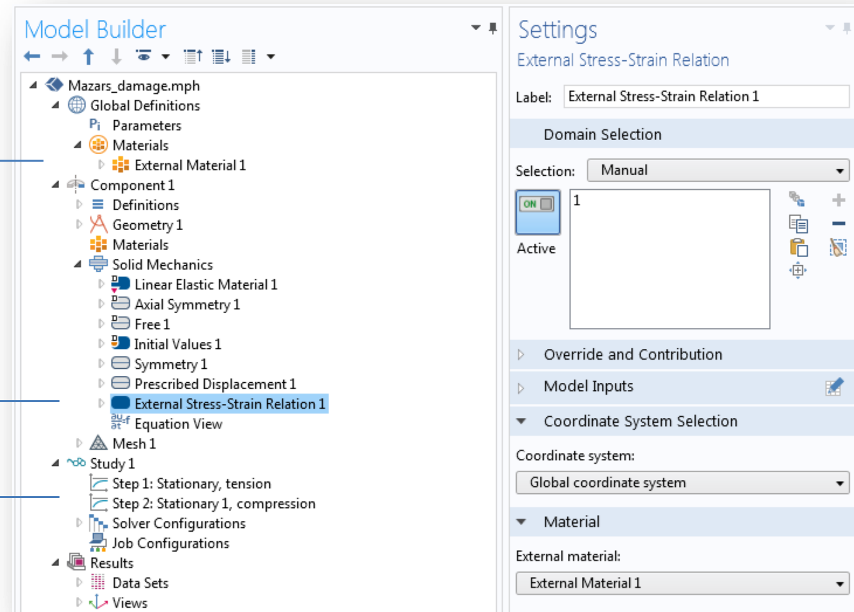
Parameters

Parameters

Name	Expression	Value	Description
para	0[m]	0 m	Parameter to control axial displacement
E0	3e10	3E10	Undamaged Young's module
nu0	0.2	0.2	Undamaged Poisson's ratio
kappa0	1e-4	1E-4	Mazars' parameter: initial damage threshold
At	1	1	Mazars' parameter At
Bt	1.5e4	15000	Mazars' parameter Bt
Ac	1.2	1.2	Mazars' parameter Ac
Bc	1.5e3	1500	Mazars' parameter Bc

Adding an External Material

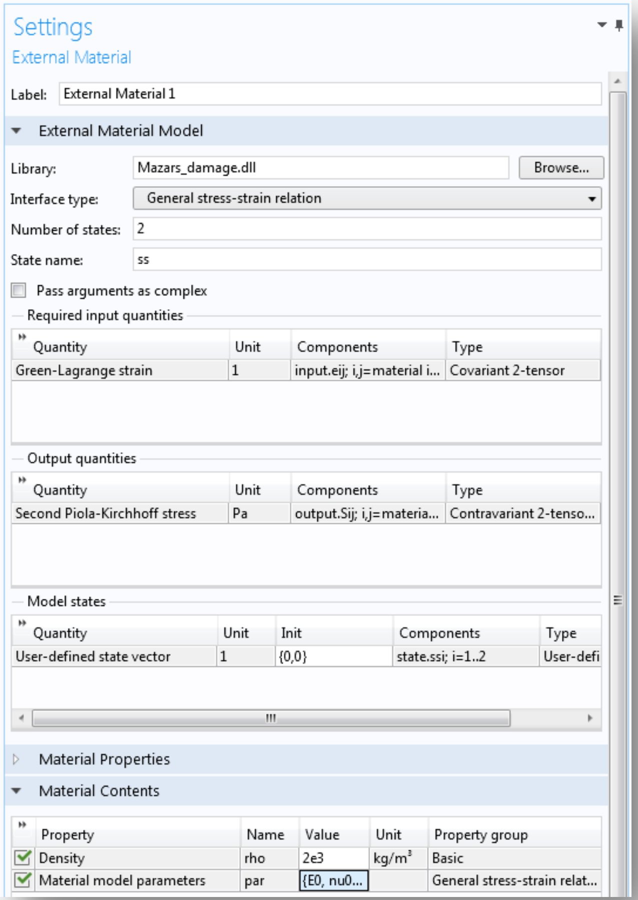
- Right-click the **Materials** node, add an **External Material** node
- Right-click **Solid Mechanics**, under **Material Models**, select **External Stress-Strain Relation**
- Add two Stationary Steps in the Study node to solve for tensile and compressive deformation



COMSOL

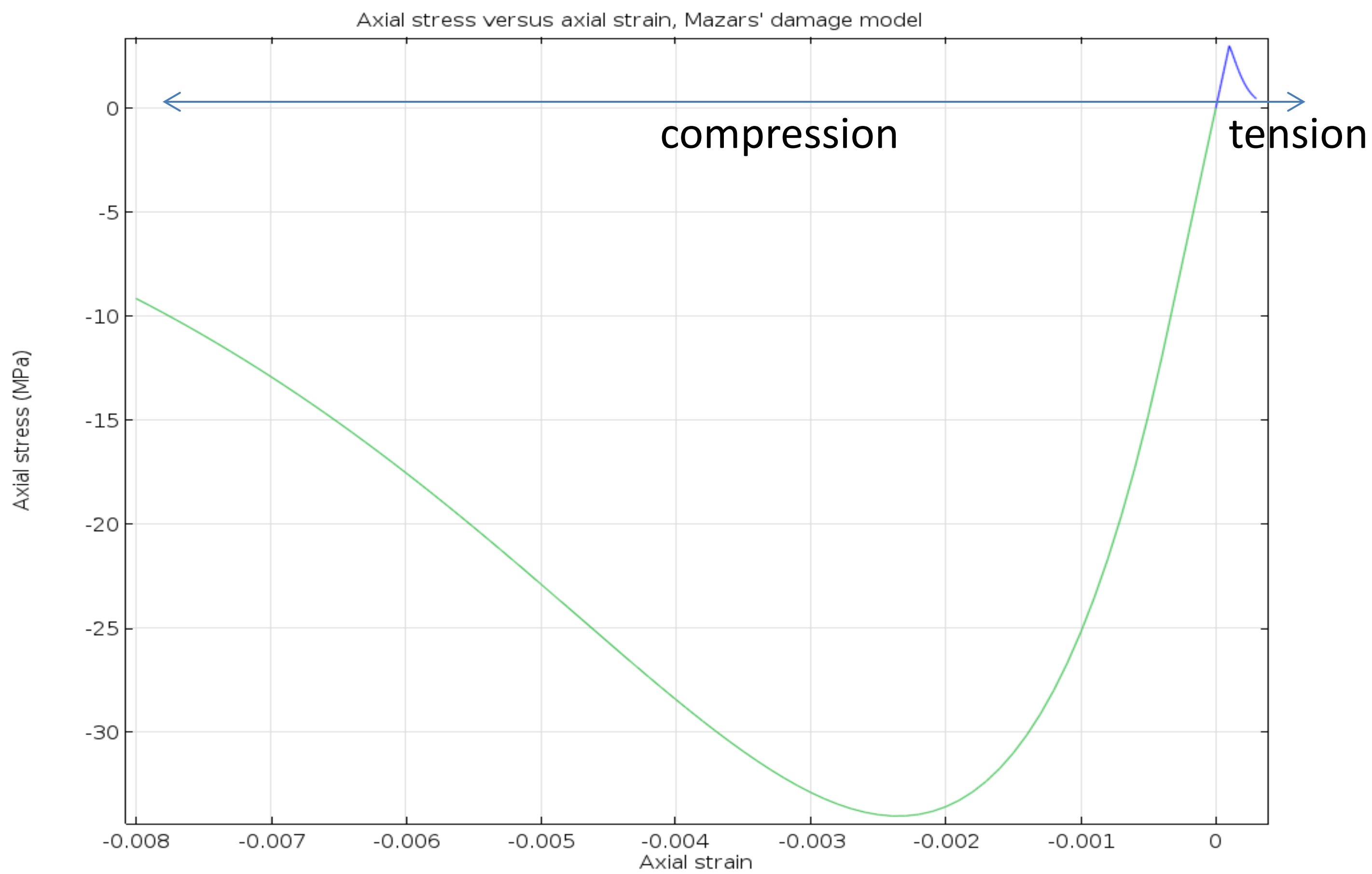
Settings for External Material

- Select Mazars_damage.dll. Not necessary to add the full path if the .dll file is located in the same folder as the .mph file
- Use General stress-strain relation
- Two states needed in this example, κ and d.
- Add Young's modulus and Poisson's ratio, and other material inputs, use brackets and commas to separate inputs



COMSOL

Results



COMSOL