

# Using External Material Functions in the Structural Mechanics Module

# 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.
- By writing a wrapper function in C code, you can also use material functions written in another programming language.
- This makes it possible to program your own material models and distribute such models as add-ons.
- Available with the
  - AC/DC Module (2D magnetics available without the AC/DC Module)
  - Structural Mechanics Module
  - MEMS modules
- 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

# External Materials

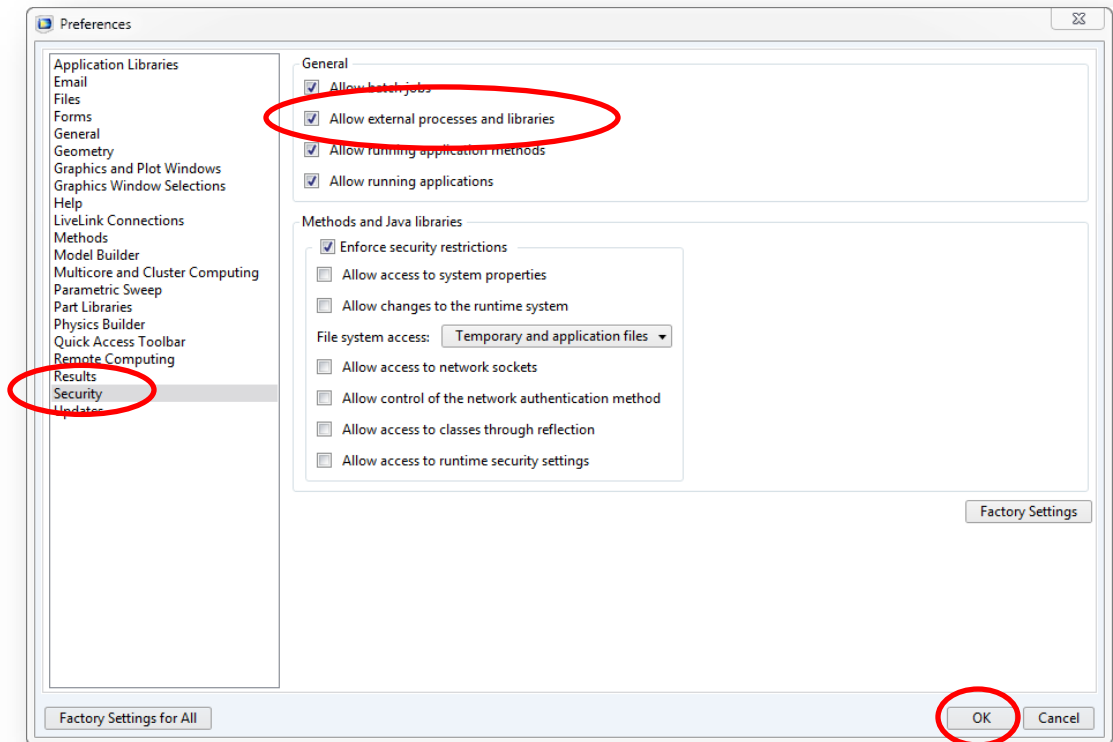
- The external material model is implemented as a C-function with a certain calling convention, compiled and linked to create dynamically linked libraries that can be called from a material node in the Model Builder at runtime.

```
1  /** Interface to an isotropic linear elastic solid with two parameters E and nu.
2  *   Example code implements linear elastic behaviour. */
3
4  /** You are allowed to use, modify, and publish this External Material File and your modifications of it subject
5  *   to the terms and conditions of the COMSOL Software License Agreement (www.comsol.com/sla). */
6
7  /** Copyright © 2015 by COMSOL. */
8
9  #include <math.h>
10 #include <stdlib.h>
11 #include <string.h>
12 #include <stdio.h>
13 #ifdef _MSC_VER
14 #define EXPORT __declspec(dllexport)
15 #else
16 #define EXPORT
17 #endif
18
19 EXPORT int eval(double *e,          // Input: Green-Lagrange strain tensor components in Voigt order (xx,yy,zz,yz,zx,xy)
20               double *s,          // Output: Second Piola-Kirchhoff stress components in Voigt order (xx,yy,zz,yz,zx,xy)
21               double *D,          // Output: Jacobian of stress with respect to strain, 6-by-6 matrix in row-major order
22               int *nPar,          // Input: Number of material model parameters, scalar
23               double *par,        // Input: Parameters: par[0] = E, par[1] = nu
24               int *nStates,       // Input: Number of states, scalar
25               double *states) {   // States, nStates-vector
26
27     // Check inputs
28     if (nPar[0] != 2)             // only two parameters needed, E and nu
29         return 1;                // error code 1 = "Wrong number of parameters"
30     if (nStates[0] != 0)          // simple linear elastic material, no states needed
31         return 2;                // error code 2 = "Wrong number of states"
```

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

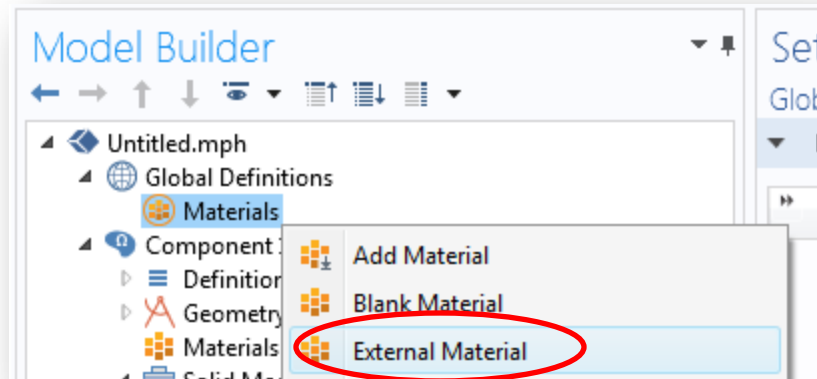
# Allowing External Processes and Libraries

- 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



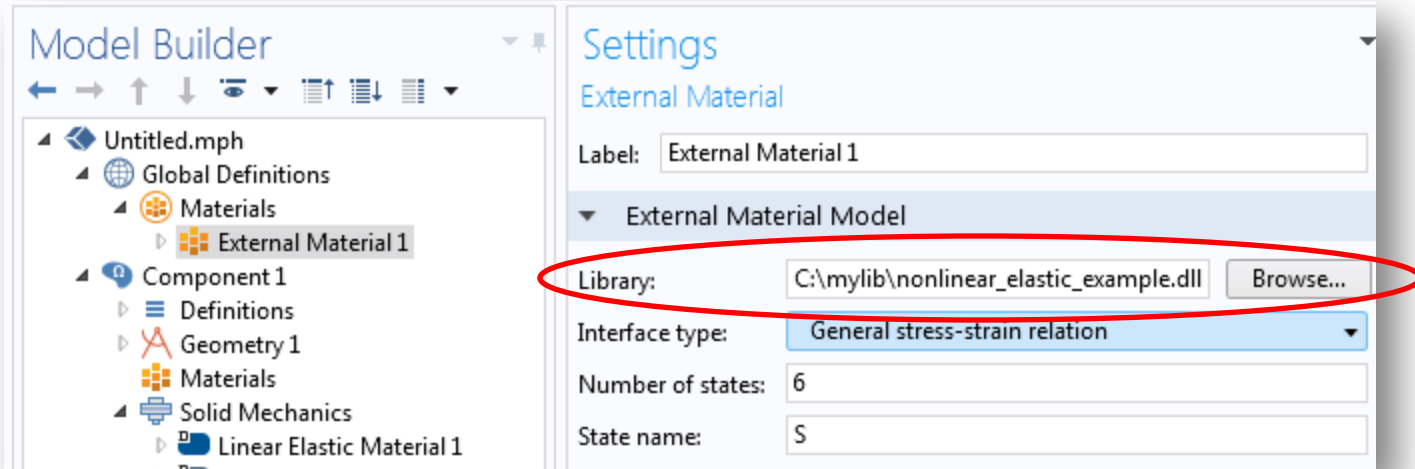
# The External Material Node

- The **External Material** node is only available under the **Global Definitions>Materials** node, not under **Materials** inside Components



# Referencing a Shared Library file

- Enter a **Library** path and name (the complete network path), or click **Browse** to locate a library to import.
- Depending on the platform, the library can be a .dll (Windows®), .so (Linux™), or .dylib (OS X) file.
- Select the **Interface type**, depending on your external library implementation.

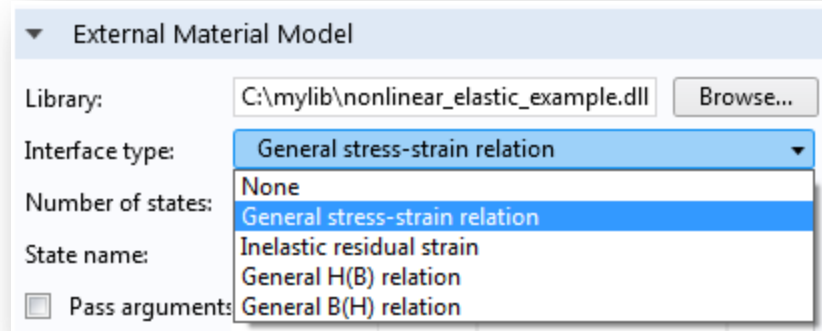


# Interface Types

- The implementation contains four different built-in **Interface types**, or *external material sockets* :

- General stress-strain relation
- Inelastic residual strain
- General H(B) relation\*
- General B(H) relation\*

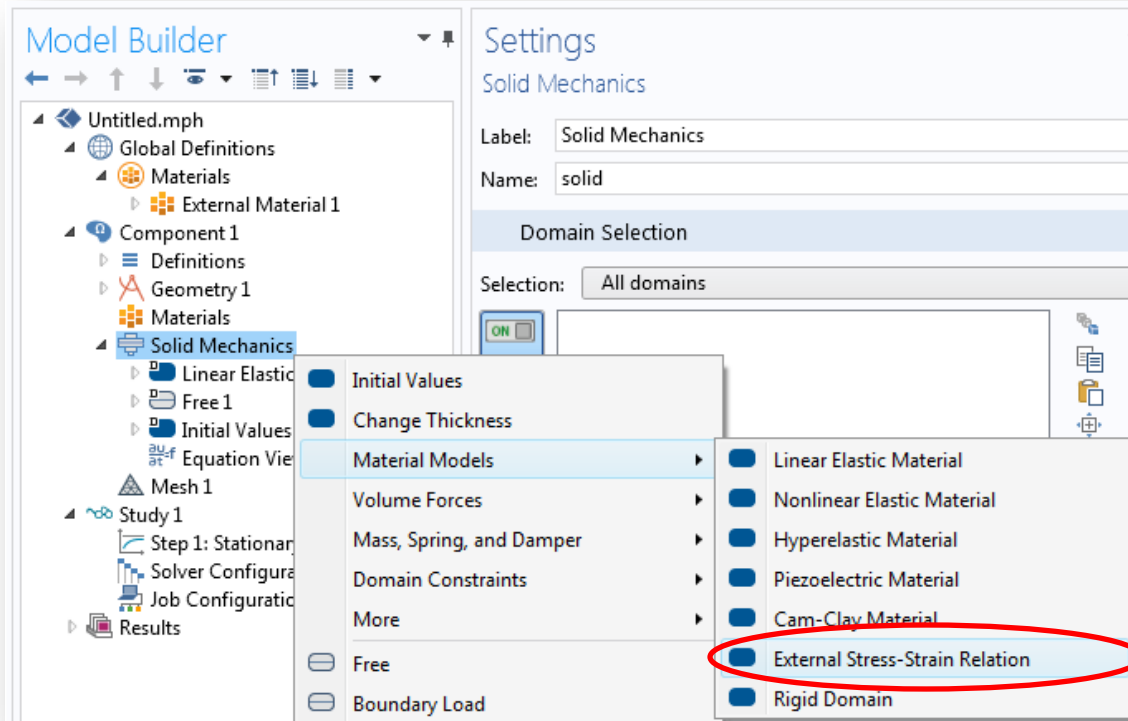
\* For magnetic materials in AC/DC Module



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

# External Materials in Solid Mechanics

- Add your external material to the domain in the same way you add any of the built-in material models.





# Selecting the External Material

- Select your material model from the list
- The **Include geometric nonlinearity** option will be selected and grayed-out in the study step

The image displays the COMSOL Model Builder interface with two panels highlighted by red circles and arrows.

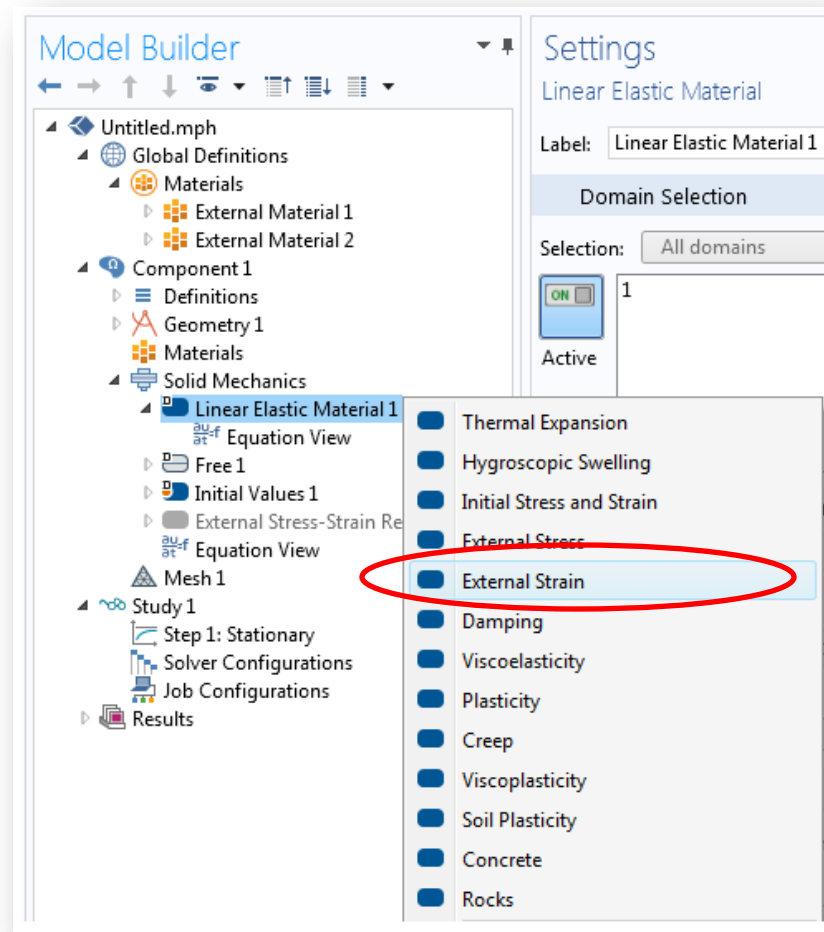
**Model Builder Panel:** The tree view on the left shows the model structure. Under 'Materials', 'External Material 1' and 'External Material 2' are circled in red. Under 'Component 1' > 'Solid Mechanics', 'External Stress-Strain Relation 1' is highlighted with a red box and a red arrow pointing to the 'Settings' panel.

**Settings Panel (External Stress-Strain Relation):** The right panel shows the settings for 'External Stress-Strain Relation 1'. The 'Material' section is expanded, and the 'External material' dropdown menu is circled in red. The list includes 'External Material 1', 'None', 'External Material 1', and 'External Material 2'. The first 'External Material 1' is selected.

**Settings Panel (Stationary):** A smaller panel on the left shows the 'Stationary' study settings. The 'Include geometric nonlinearity' checkbox is checked and circled in red. A red arrow points from this checkbox to the 'External Stress-Strain Relation 1' entry in the Model Builder tree.

# Inelastic Residual Strain and External Strain

- For the **Inelastic residual strain** interface type (selected in the External Material node), add an **External Strain** contribution to a **Linear Elastic Material** in the same way as adding any of the built-in inelastic strains contributions.

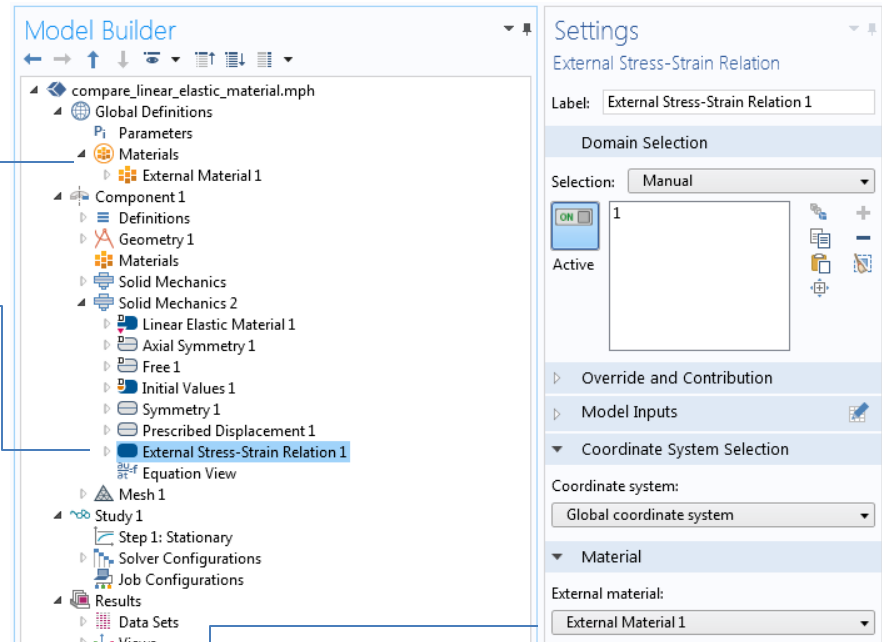


# Example

- Compare a linear elastic material written in C to the built-in Linear elastic material
- Use compiled library `linear_elastic.dll`
- Add two Solid Mechanics interfaces on a simple 2D axisymmetric geometry
- Uniaxial tensile test, 5 % axial strain
- Material parameters:  $E=2e9$  Pa,  $\nu=0.3$

# 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**
- In the Settings for External Stress-Strain Relation, select the domains to use the external material, under the **External material** list, select **External Material 1**



# Settings for External Material

- Select your library. 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
- No need for states in this example
- Add Young's modulus and Poisson's ratio, use brackets and commas to separate inputs {2e9,0.3}

Settings  
External Material

Label: External Material 1

External Material Model

Library: linear\_elastic.dll Browse...

Interface type: General stress-strain relation

Number of states: 0

State name: S

☐ Pass arguments as complex

Required input quantities

Quantity	Unit	Components	Type
Green-Lagrange strain	1	input.eij; i,j=material...	Covariant 2-ten

Output quantities

Quantity	Unit	Components	Type
Second Piola-Kirchhoff stress	Pa	output.Sij; i,j=materi...	Contravariant 2

Model states

Quantity	Unit	Init	Components
User-defined state vector	1	{}	state.Si; i=1..0

Material Properties

Material Contents

Property	Name	Value	Unit	Property grou
<input checked="" type="checkbox"/> Density	rho	8e3	kg/m³	Basic
<input checked="" type="checkbox"/> Material model parameters	par	{2e9, 0.3}		General stress-

# Results

