Using External Material Functions in the Structural Mechanics Module

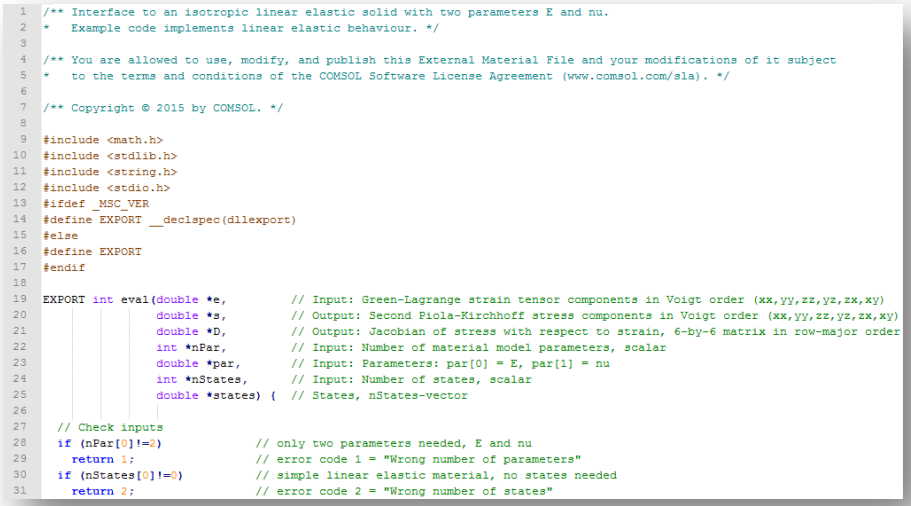
COMSOL

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.
- 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.



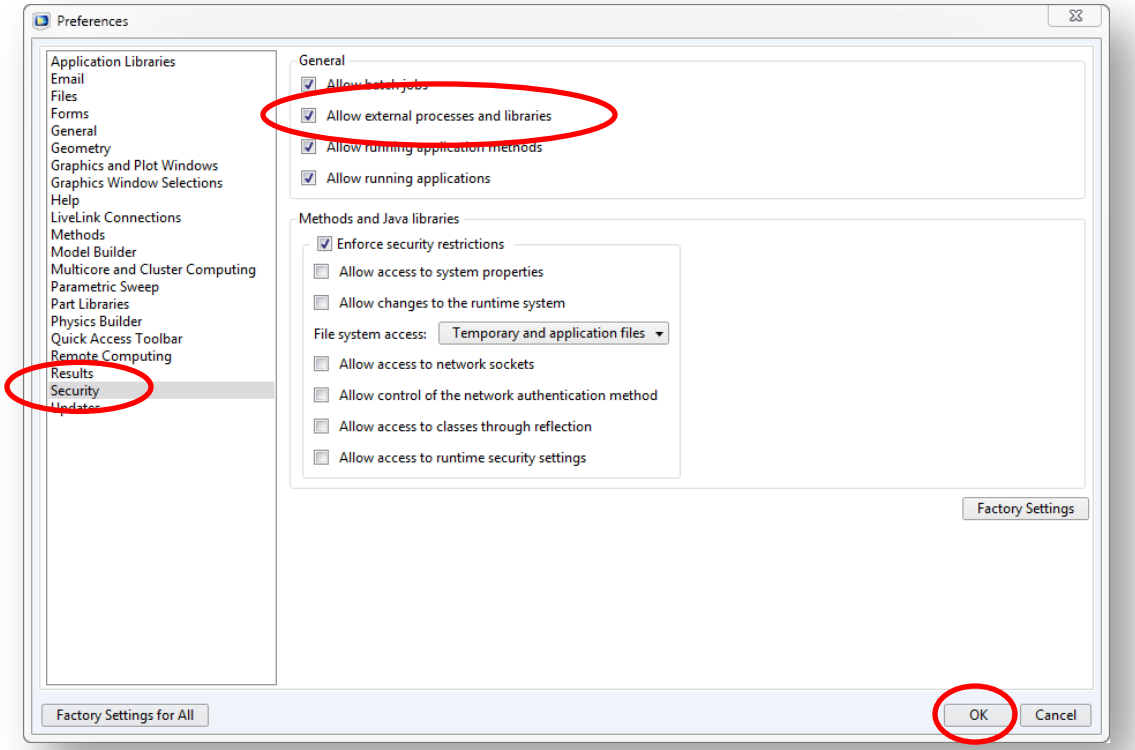
```c
/** Interface to an isotropic linear elastic solid with two parameters E and nu.
*   Example code implements linear elastic behaviour. */

/** You are allowed to use, modify, and publish this External Material File and your modifications of it subject
*   to the terms and conditions of the COMSOL Software License Agreement (www.comsol.com/sla). */

/** Copyright © 2015 by COMSOL. */

#include <math.h>
#include <stdlib.h>
#include <string.h>
#include <stdio.h>
#ifdef _MSC_VER
#define EXPORT __declspec(dllexport)
#else
#define EXPORT
#endif

EXPORT int eval(double *e,         // Input: Green-Lagrange strain tensor components in Voigt order (xx,yy,zz,yz,zx,xy)
                double *s,         // Output: Second Piola-Kirchhoff stress components in Voigt order (xx,yy,zz,yz,zx,xy)
                double *D,         // Output: Jacobian of stress with respect to strain, 6-by-6 matrix in row-major order
                int *nPar,         // Input: Number of material model parameters, scalar
                double *par,       // Input: Parameters: par[0] = E, par[1] = nu
                int *nStates,      // Input: Number of states, scalar
                double *states) {  // States, nStates-vector

  // Check inputs
  if (nPar[0]!=2)             // only two parameters needed, E and nu
    return 1;                 // error code 1 = "Wrong number of parameters"
  if (nStates[0]!=0)          // simple linear elastic material, no states needed
    return 2;                 // error code 2 = "Wrong number of states"
```

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

COMSOL

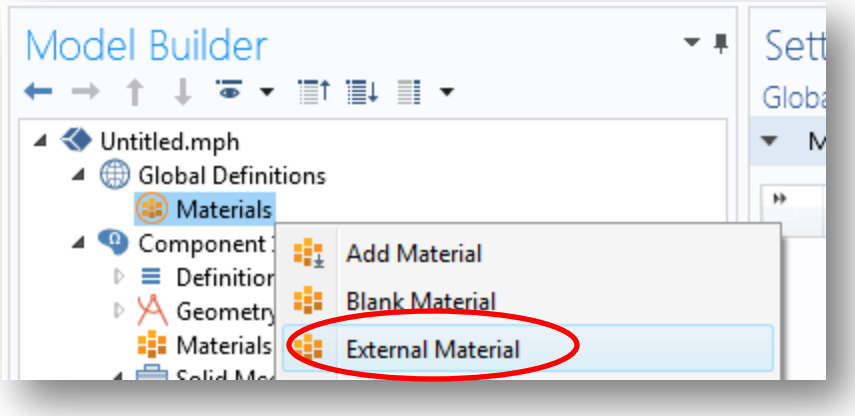
Allowing External Processes and Libraries

- 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



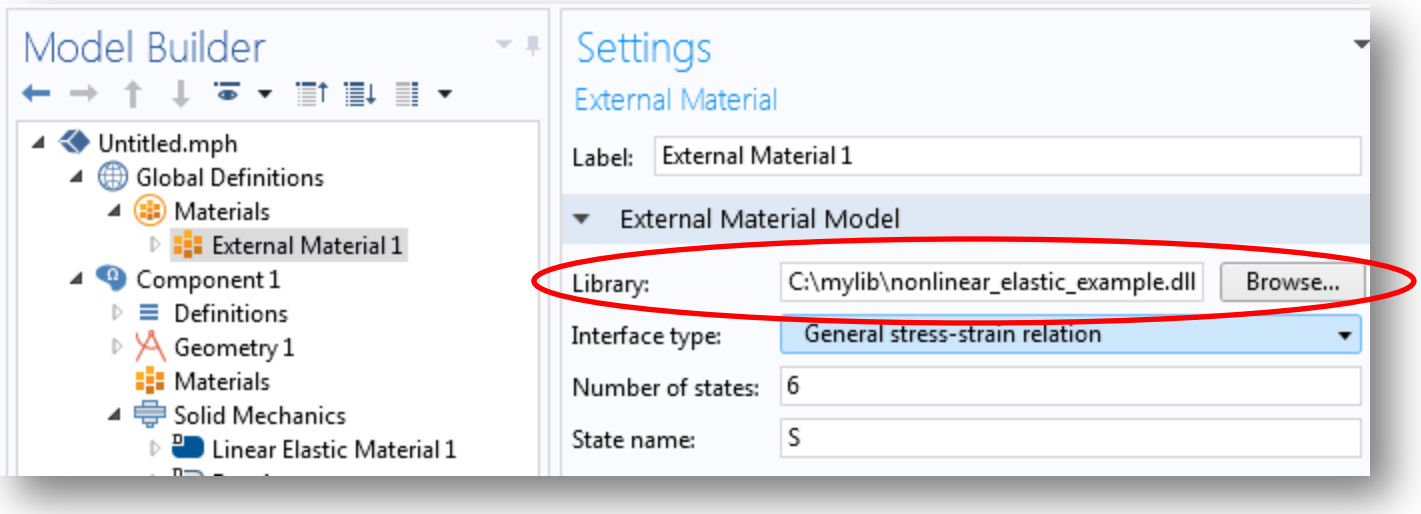
The External Material Node

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



Referencing a Shared Library file

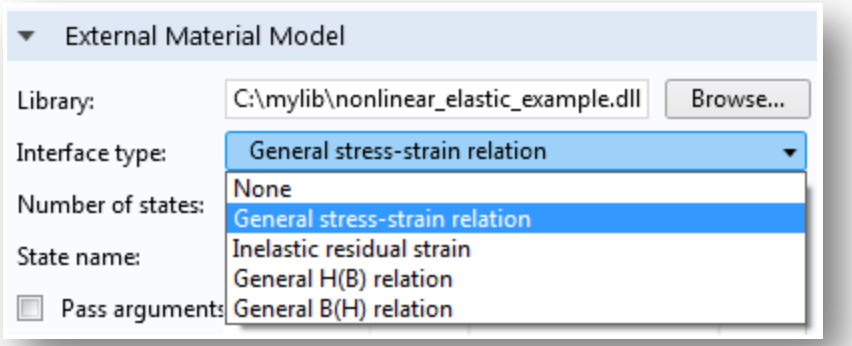
- Enter a **Library** path and name (the complete network path), or click **Browse** to locate a library to import.
- Depending on the platform, the library can be a .dll (Windows®), .so (Linux™, or .dylib (OS X) file.
- Select the **Interface type**, depending on your external library implementation.



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*

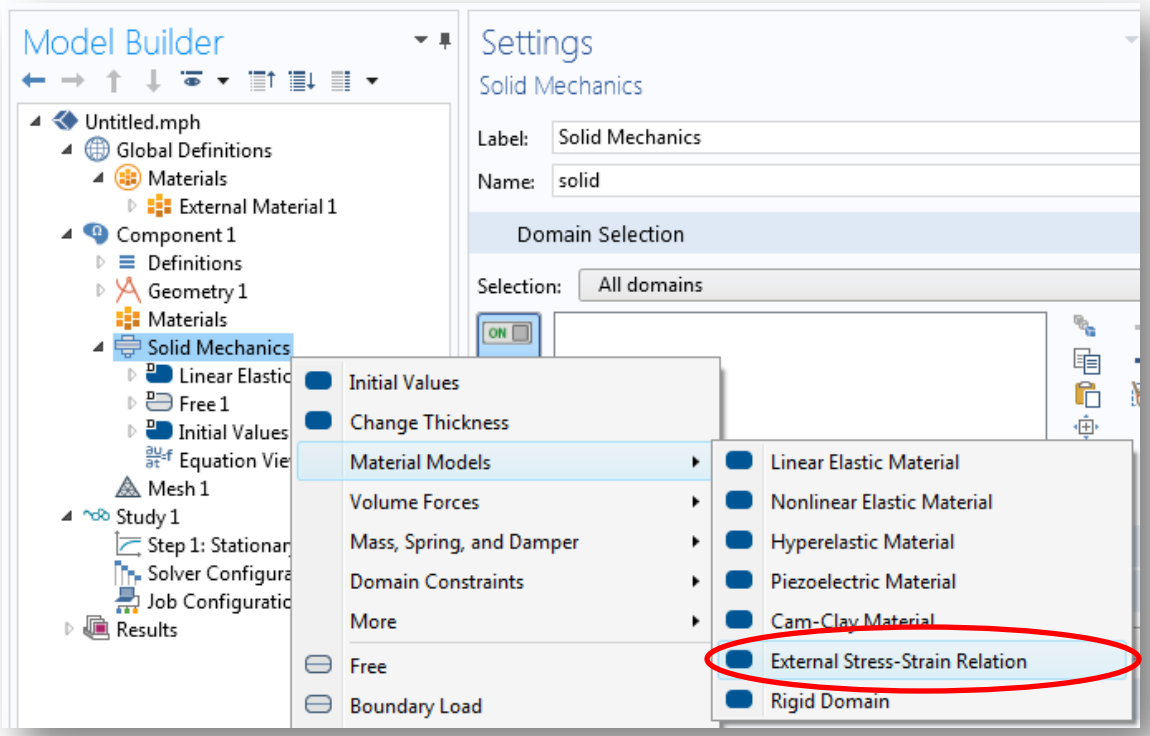
    * For magnetic materials in AC/DC Module



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

External Materials in Solid Mechanics

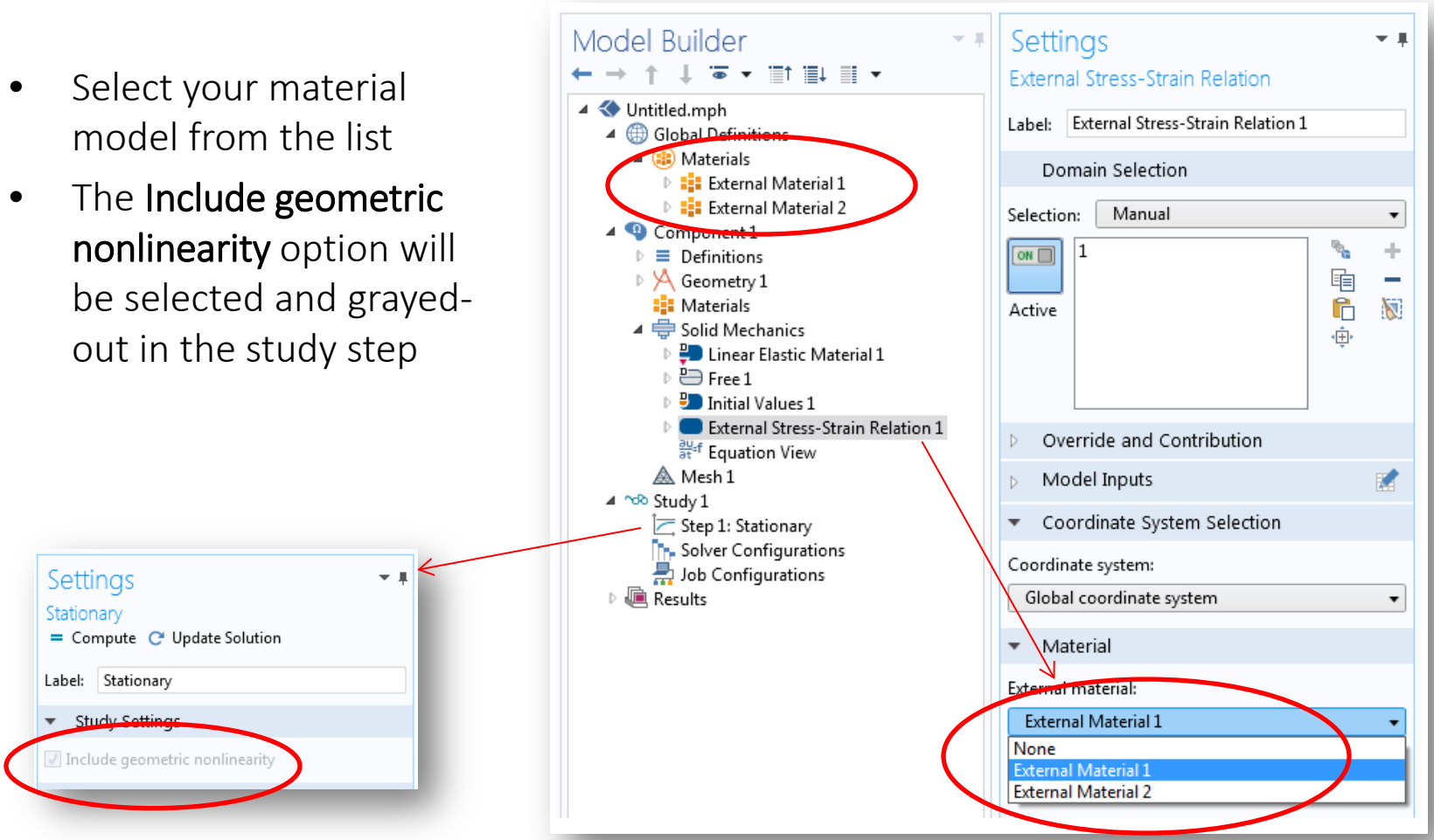
- Add your external material to the domain in the same way you add any of the built-in material models.



COMSOL

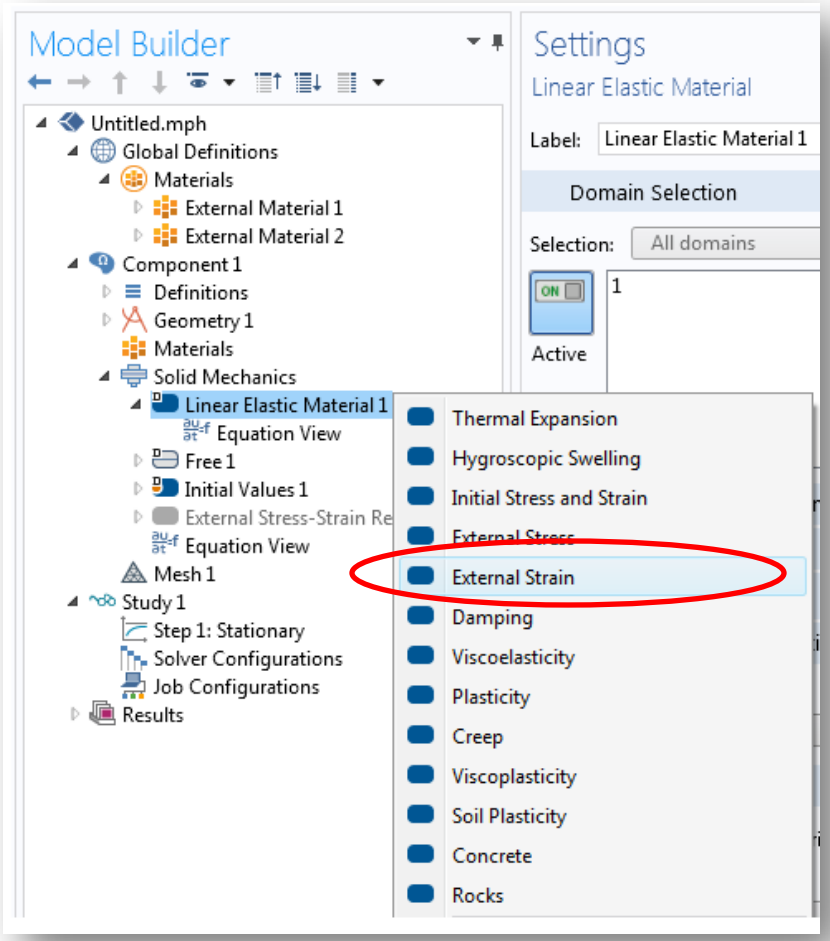
Selecting the External Material

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



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.

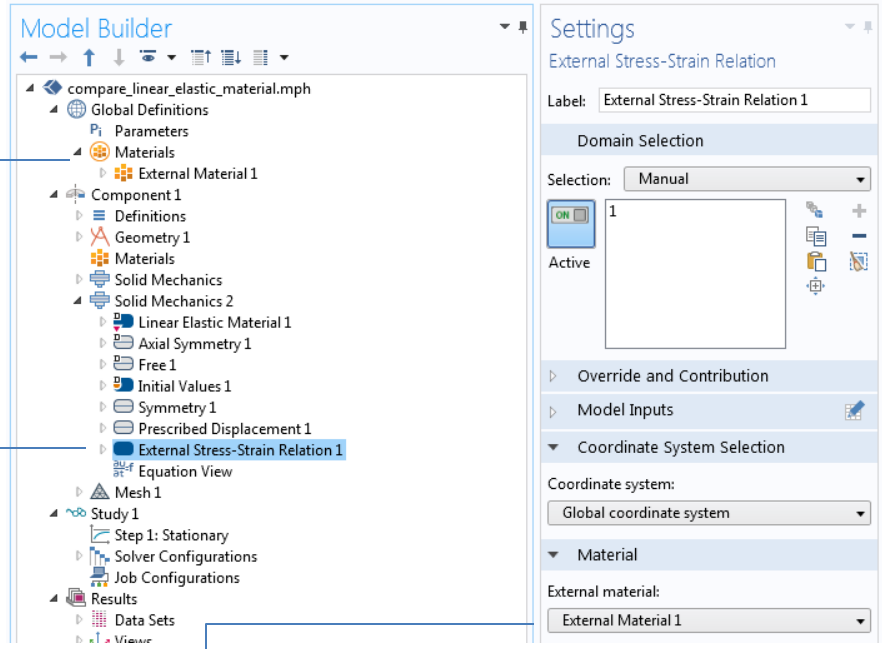


Example

- 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, ν=0.3

Adding an External Material

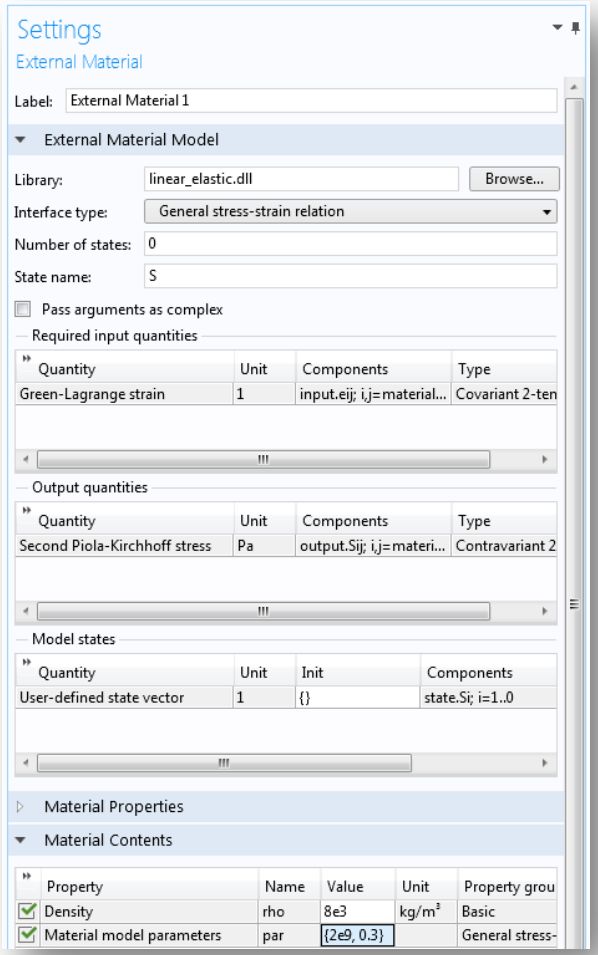
- Right-click the **Materials** node, add an **External Material** node
- Right-click **Solid Mechanics**, under **Material Models**, select **External Stress-Strain Relation**
- In the Settings for External Stress-Strain Relation, select the domains to use the external material, under the **External material** list, select External Material 1



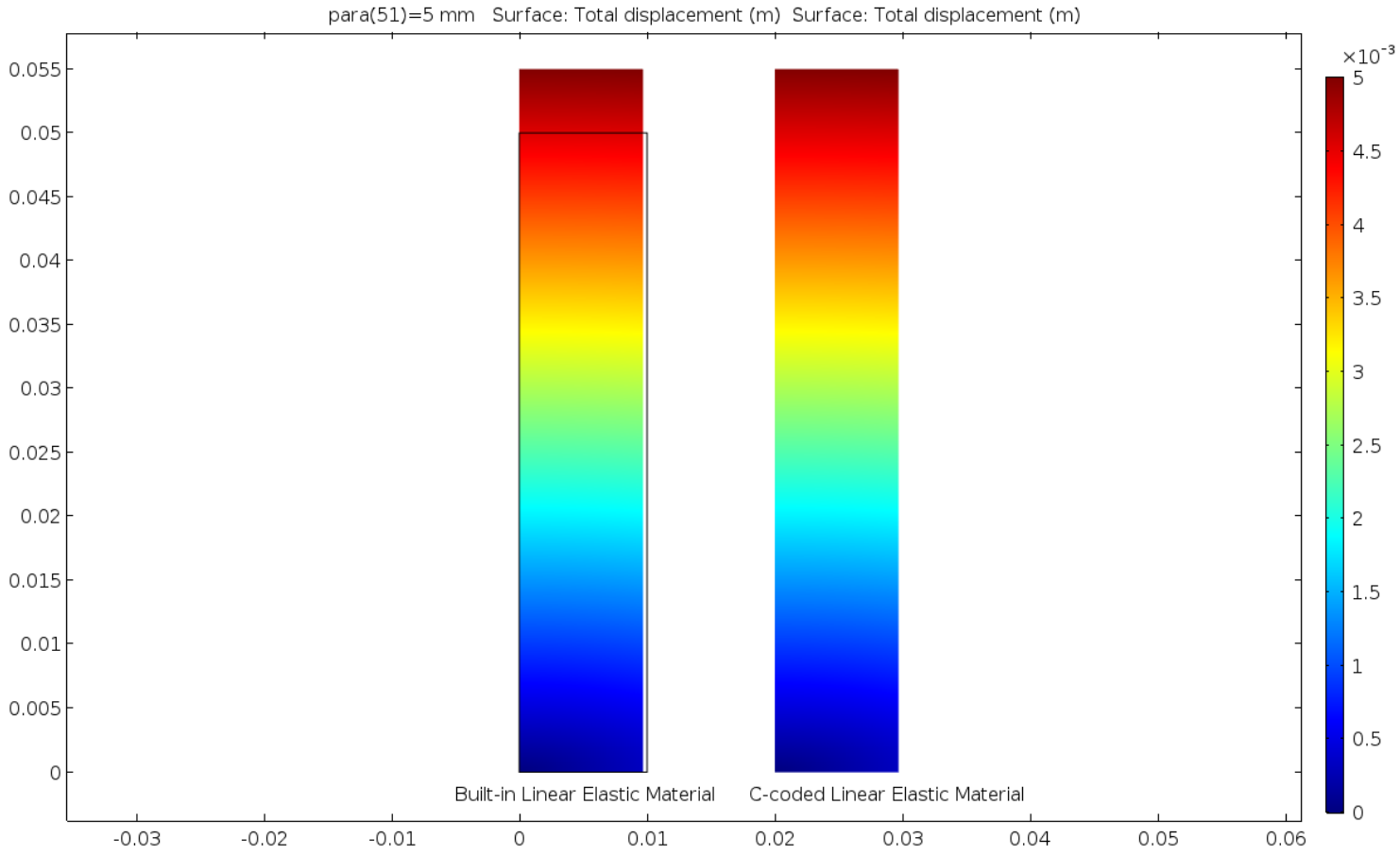
COMSOL

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}



Results



COMSOL