Material Designer Nonlinear Materials (Beta)

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2019 R3
August 2019

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2019 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

DISCLAIMER

This document describes beta features.

Beta documentation and features are considered unreleased and have not been fully tested nor fully validated. The results are not guaranteed by ANSYS, Inc. (ANSYS) to be correct. You assume the risk of using beta features and beta documentation.

At its discretion, ANSYS, Inc. (ANSYS) may release, change, or withdraw beta features in future revisions.

Beta features are not subject to the ANSYS Class 3 error reporting system. ANSYS makes no commitment to resolve defects reported against beta features; however, your feedback will help us improve the quality of the product.

ANSYS does not guarantee that input files used with beta features will run successfully from version to version of the software, nor with the final released version of the features. You may need to modify the input files before running them on other versions.

BETA

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

BETA

BETA

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

BETA

BETA

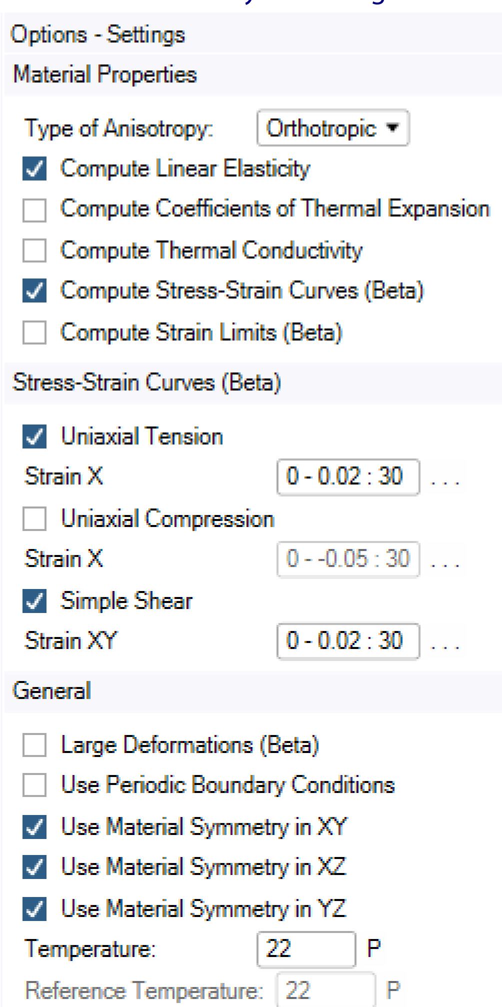
Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Using Nonlinear Material Properties in Material Designer

You can compute stress-strain curves for RVEs with nonlinear constituent materials, simulating uniaxial tension, uniaxial compression, and shear test cases. Both hyperelastic and elasto-plastic material models are supported for the constituents. This document explains how to use the nonlinear material properties, which are provided in the 2019 R3 release as a Beta feature. See this section for information about how to activate Material Designer Beta features.

1.1. Nonlinear Material Analysis Settings

There are new Analysis Settings related to the stress/strain curves:



- **Compute Stress-Strain Curves**: Compute stress-strain curves based on one or more numerical tests.
- **Stress-Strain**: Define which tests are carried out. You can activate uniaxial tension, uniaxial compression, and simple shear test cases.

  For each test case, you can specify for which values of strain the stresses are evaluated.

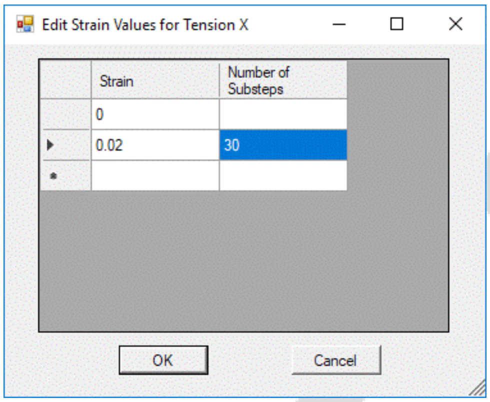
Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For instance, 0 - 0.02 : 30 means that an analysis with 30 substeps is run, where the strain is increased from 0 to 0.02. You can also use a comma separated list; for instance 0.005, 0.01, 0.0015, 0.02.

Note

- You can also specify a loading and unloading; for instance by stating 0 - 0.02 : 30, 0.02 - 0 : 30. Or you can even have more complicated loading paths (including cyclic loading).
- If you use a comma as the decimal delimiter (per the number format settings in Windows), you nevertheless need to input the numbers here with a dot as the decimal delimiter.
- Some of the tests are disabled when material symmetries are activated.

Press the **...** button to edit the strain values more easily using a dialog:



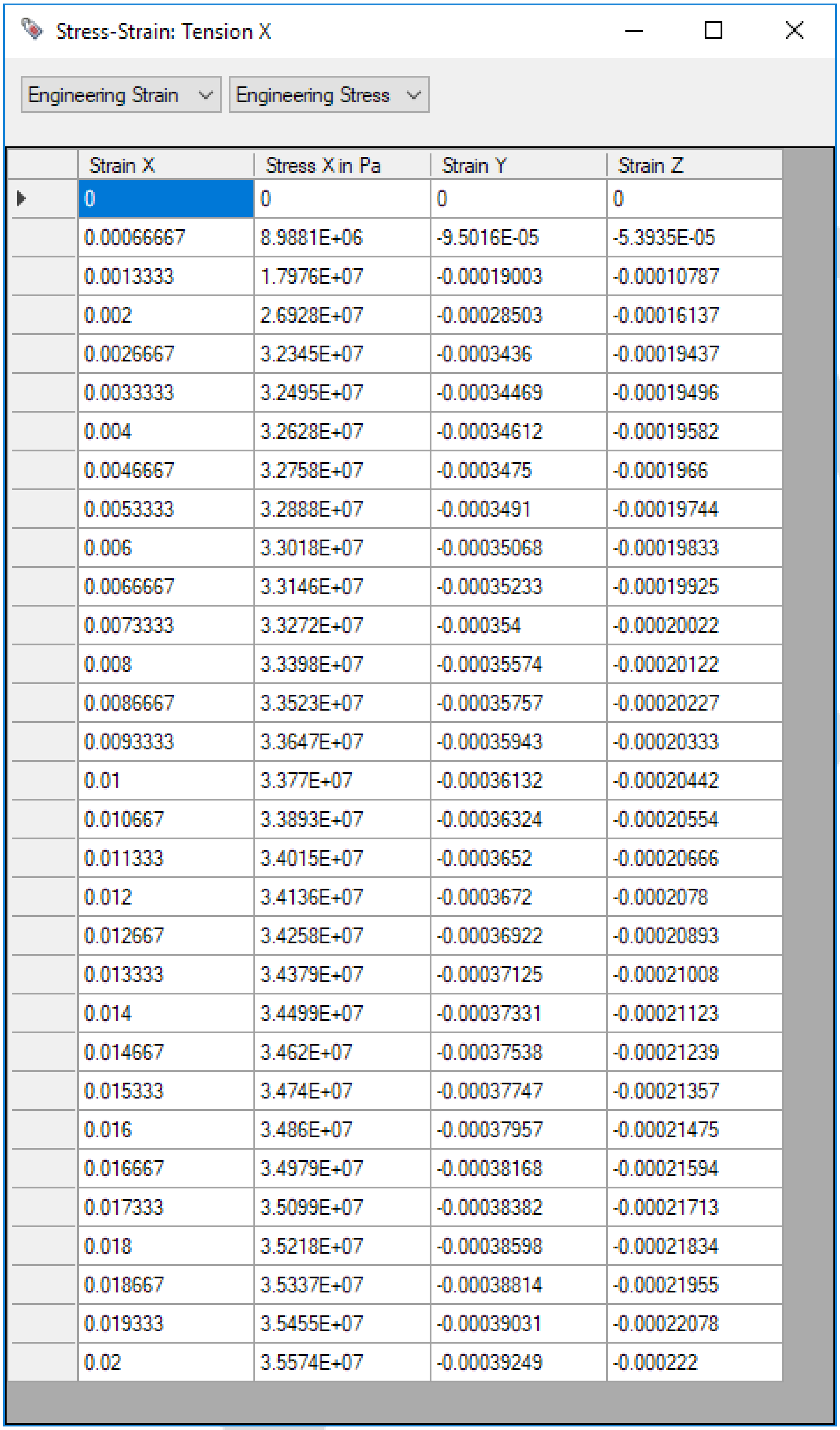
- **Large Deformations**: Activate large deformations. Input material properties based on strain-stress curves are interpreted as true strain vs. true stress (see Large Deformation in the Mechanical APDL Material Reference). You will need to activate large deformations if the RVE contains hyperelastic materials.

1.2. Nonlinear Material Results

Stress-Strain Curves is available under the Constant and Raw Variable Results Table. Click the icon to see the data of the corresponding stress-strain curve:

BETA

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Strain X	Stress X in Pa	Strain Y	Strain Z
0	0	0	0
0.00066667	8.9881E+06	-9.5016E-05	-5.3935E-05
0.0013333	1.7976E+07	-0.00019003	-0.00010787
0.002	2.6928E+07	-0.00028503	-0.00016137
0.0026667	3.2345E+07	-0.0003436	-0.00019437
0.0033333	3.2495E+07	-0.00034469	-0.00019496
0.004	3.2628E+07	-0.00034612	-0.00019582
0.0046667	3.2758E+07	-0.0003475	-0.0001966
0.0053333	3.2888E+07	-0.0003491	-0.00019744
0.006	3.3018E+07	-0.00035068	-0.00019833
0.0066667	3.3146E+07	-0.00035233	-0.00019925
0.0073333	3.3272E+07	-0.000354	-0.00020022
0.008	3.3398E+07	-0.00035574	-0.00020122
0.0086667	3.3523E+07	-0.00035757	-0.00020227
0.0093333	3.3647E+07	-0.00035943	-0.00020333
0.01	3.377E+07	-0.00036132	-0.00020442
0.010667	3.3893E+07	-0.00036324	-0.00020554
0.011333	3.4015E+07	-0.0003652	-0.00020666
0.012	3.4136E+07	-0.0003672	-0.0002078
0.012667	3.4258E+07	-0.00036922	-0.00020893
0.013333	3.4379E+07	-0.00037125	-0.00021008
0.014	3.4499E+07	-0.00037331	-0.00021123
0.014667	3.462E+07	-0.00037538	-0.00021239
0.015333	3.474E+07	-0.00037747	-0.00021357
0.016	3.486E+07	-0.00037957	-0.00021475
0.016667	3.4979E+07	-0.00038168	-0.00021594
0.017333	3.5099E+07	-0.00038382	-0.00021713
0.018	3.5218E+07	-0.00038598	-0.00021834
0.018667	3.5337E+07	-0.00038814	-0.00021955
0.019333	3.5455E+07	-0.00039031	-0.00022078
0.02	3.5574E+07	-0.00039249	-0.000222

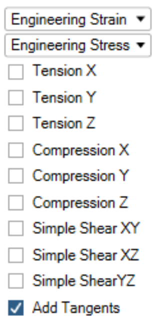
You can switch between Engineering Strain/Stress and True Strain/Stress.

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3. Stress-Strain Charts

You can generate stress-strain charts for Constant or Variable material evaluations. To do so, right-click Constant Material Evaluation or Variable Material Evaluation in the outline window and select Add Stress-Strain Chart.

This adds a node in the outline and shows the following options:



Strain: Choose whether to plot the Engineering Strain or the True Strain.

Stress: Choose whether to plot the Engineering Stress or the True Stress.

Load Cases: Choose for which load cases the stress-strain curves are shown.

Add Tangents (only available for constant material): Choose to add tangents that would correspond to a linear elastic material behavior (based on the computed engineering constants); see the dotted lines in Figure 1.1: Example of Stress-Strain Curves (p. 5).

This allows you to verify that the computed material properties are consistent with the computed stress-strain curves and allows you to see deviation from linear elastic material behavior more easily.

Note

For large strain the tangents do not necessarily represent the linear elastic material behavior. If large deformations are active, then the tangents in the true strain-true stress diagram correspond to the linear material behavior. Similarly, if large deformations are off, then the tangents in the engineering strain-engineering stress diagram correspond to the linear material behavior.

Click Complete to open a new window tab with the chart.

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: Example of Stress-Strain Curves

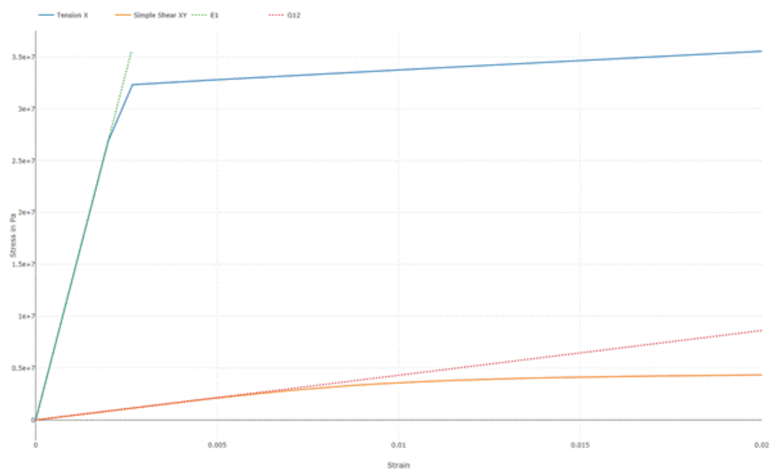
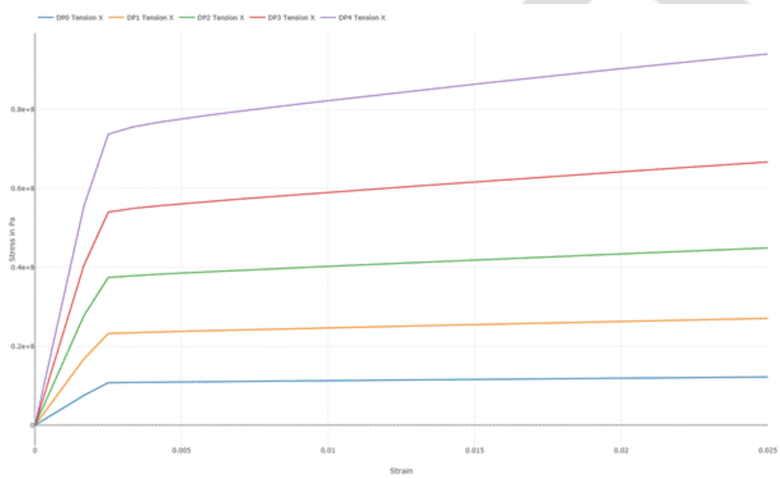


Figure 1.2: Example of Stress-Strain Curves (Variable)



1.4. Exporting Stress-Strain Curves

When you Export stress-strain curves, they are exported as Engineering Stress vs. Engineering Strain.

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BETA

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Nonlinear Material Properties Theory

In the following, we highlight how computing material properties differs for nonlinear constituent materials and show the theory behind the computation of stress-strain curves.

2.1. Orthotropic Linear-Elastic Material Properties

The computation of orthotropic linear-elastic materials is described in this section for linear constituent materials. The following considerations apply in addition when using the new beta feature.

2.1.1. Nonlinear Constituent Materials

Note that you can also compute orthotropic linear-elastic material properties if the constituent materials are nonlinear. The computed properties are then valid only in a small strain analysis.

If possible the nonlinear effects are ignored completely, similar to setting nonlinear material effects to Off in Mechanical (see Nonlinear Material Effects).

In case this is not possible (for instance, for hyperelastic materials) then the nonlinear material is used, which will lead to some errors in computing the linear elastic material behavior as the applied strain is not infinitesimal, but finite. However, the effect will be negligible in most cases.

2.1.2. Large Deformations

Note that if large deformations are active, input material properties based on strain-stress curves are interpreted as true strain vs. true stress, as discussed in Large Deformation in the Mechanical APDL Material Reference.

The material properties are corrected for the fact that true strain rather than engineering strain is the active strain measure. Similarly, the area changes are taken into account when computing material properties.

However, it is not possible to setup a "clean" shear case if large deformations are active; there are always some other macroscopic strains that are non-zero (see Polar Decomposition of a Shearing Deformation and the corresponding description). Once again, the effects will be negligible in most cases given that the applied shear is small.

2.2. Computing Stress-Strain Curves

Consider for the moment that we want to compute stress-strain curves for a tension test in X direction for a periodic RVE with a periodic mesh. The basic principle is that we specify several values for the strain in the X direction and measure the force in the X direction. After normalization, this yields the corresponding stress values.

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In contrast to the boundary conditions specified here, we allow the RVE to contract macroscopically in Y and Z direction; in other words, we allow a non-zero lateral strain ϵ_y and ϵ_z. To do so, we introduce 6 degrees of freedom u_x^{pivot}, u_y^{pivot}, u_z^{pivot}, rot_x^{pivot}, rot_y^{pivot}, and rot_z^{pivot} (via an additional node in the finite element model) and let them correspond to the macroscopic strains.

Assume that the RVE occupies the volume $[0,L_x]$ x $[0,L_y]$ x $[0,L_z]$. Then we enforce the following boundary conditions:

On the faces normal to the X-axis, enforce

$$u_x(L_x,y,z) = u_x(0,y,z) + u_x^{\text{pivot}} L_x$$
$$u_y(L_x,y,z) = u_y(0,y,z)$$
$$u_z(L_x,y,z) = u_z(0,y,z)$$

On faces normal to the Y-axis, enforce

$$u_x(x,L_y,z) = u_x(x,0,z)$$
$$u_y(x,L_y,z) = u_y(x,0,z) + u_y^{\text{pivot}} L_y$$
$$u_z(x,L_y,z) = u_z(x,0,z)$$

On faces normal to the Z-axis, enforce

$$u_x(x,y,L_z) = u_x(x,y,0)$$
$$u_y(x,y,L_z) = u_y(x,y,0)$$
$$u_z(x,y,L_z) = u_z(x,y,0) + u_z^{\text{pivot}} L_z$$

In addition to these periodicity conditions, rigid body motions must also be prevented. This is done by enforcing

$$u_x(\text{a point with } x = 0) = 0$$
$$u_y(\text{a point with } y = 0) = 0$$
$$u_z(\text{a point with } z = 0) = 0$$

The strain in X is enforced by setting

$$u_x^{\text{pivot}} = \epsilon_x$$

while u_y^{pivot}, u_z^{pivot} are unconstrained, allowing lateral contraction of the RVE.

ϵ_x is successively set to the values specified by the user.

The remaining degrees of freedom rot_x^{pivot}, rot_y^{pivot}, rot_z^{pivot} are not needed for this load case and are set to zero. They will be used in the shear load cases to introduce shear strains.

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that we aim to compute engineering strain vs. engineering stress (even in case large deformations are active). Thus, the force in the X direction is integrated and normalized by the original area L_y L_z to obtain σ_x.

The lateral strains can be extracted from the degrees of freedom of the pivot node:

$$\epsilon_y = u_y^{\text{pivot}}$$
$$\epsilon_z = u_z^{\text{pivot}}$$

In case true (logarithmic) strains are needed, you can compute them via

$$\epsilon_x^l = \log(1 + \epsilon_x)$$

To compute true stresses, one needs to compensate the stress for area changes due to contraction, which leads to

$$\sigma_x^{\text{true}} = \frac{\sigma_x}{(1 + \epsilon_y)(1 + \epsilon_z)}$$

The boundary conditions for the other test cases will be specified in the following.

Note that the stress-strain computation in Material Designer does not take into account (microscopic) damage, such as pull-out of fibers, micro-cracks, delamination, nonlinear interfaces, etc. If these effects are important, then Material Designer is not suitable to model and simulate this RVE.

Further note that it is not possible to setup a "clean" shear case if large deformations are active; in other words, there are always some other macroscopic strains that are non-zero (see Polar Decomposition of a Shearing Deformation and the corresponding description). For stress-strain curves, there will in general be a non-negligible effect given that the applied strains can be large.

Note that Material Designer is limited to rate-independent material behavior. You can use constituent materials that show a hyperelastic or a rate-independent plastic behavior. Rate-dependent material properties, like viscoplasticity, viscoelasticity, and creep are not supported.

2.2.1. Periodic Boundary Conditions

For completeness, the boundary conditions used in the different load cases for the stress-strain curves are presented here as well.

In each load case, one of the quantities $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}$, and γ_{xz} is successively set to the values specified by the user.

In tension and compression cases, the remaining two normal strains of $\epsilon_x, \epsilon_y, \epsilon_z$ are left free, while γ_{xy}, γ_{yz}, and γ_{xz} are set to zero.

In the simple shear cases, all quantities $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}$, and γ_{xz} except the one specified by the user are set to zero.

Then there is a one-to-one correspondence between the macroscopic strains and the degrees of freedom of the pivot node:

$$\begin{aligned} u_x^{\text{pivot}} &= \epsilon_x \quad & rot_x^{\text{pivot}} &= \gamma_{yz} \\ u_y^{\text{pivot}} &= \epsilon_y \quad & rot_y^{\text{pivot}} &= \gamma_{xz} \\ u_z^{\text{pivot}} &= \epsilon_z \quad & rot_z^{\text{pivot}} &= \gamma_{xy} \end{aligned}$$

With these macroscopic strains in place, the boundary conditions are defined as follows:

On the faces normal to the X-axis, enforce

$$\begin{aligned} u_x(L_x, y, z) &= u_x(0, y, z) + u_x^{\text{pivot}} L_x \\ u_y(L_x, y, z) &= u_y(0, y, z) + rot_z^{\text{pivot}} L_x \\ u_z(L_x, y, z) &= u_z(0, y, z) + rot_y^{\text{pivot}} L_x \end{aligned}$$

On the faces normal to the Y-axis, enforce

$$\begin{aligned} u_x(x, L_y, z) &= u_x(x, 0, z) \\ u_y(x, L_y, z) &= u_y(x, 0, z) + u_y^{\text{pivot}} L_y \\ u_z(x, L_y, z) &= u_z(x, 0, z) + rot_x^{\text{pivot}} L_y \end{aligned}$$

On the faces normal to the Z-axis, enforce

$$\begin{aligned} u_x(x, y, L_z) &= u_x(x, y, 0) \\ u_y(x, y, L_z) &= u_y(x, y, 0) \\ u_z(x, y, L_z) &= u_z(x, y, 0) + u_z^{\text{pivot}} L_z \end{aligned}$$

To avoid rigid body motions, enforce

$$\begin{aligned} &u_x(\text{a point with } x = 0) = 0 \\ &u_y(\text{a point with } y = 0) = 0 \\ &u_z(\text{a point with } z = 0) = 0 \end{aligned}$$

Note that the compression and tension cases are the same, except that you are expected to specify different strain values.

2.2.2. Non-Periodic Boundary Conditions

For the tension and compression tests, one quantity of ϵ_x, ϵ_y, and ϵ_z is nonzero, the other two normal strain are left free. All the shear strains ϵ_{xy}, ϵ_{xz}, and ϵ_{yz} are set to zero.

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once again there is a one-to-one correspondence between the macroscopic strains and the degrees of freedom of the pivot node:

$$
\begin{aligned}
u_x^{\text{pivot}} &= \epsilon_x \quad & rot_x^{\text{pivot}} &= \gamma_{yz} \\
u_y^{\text{pivot}} &= \epsilon_y \quad & rot_y^{\text{pivot}} &= \gamma_{xz} \\
u_z^{\text{pivot}} &= \epsilon_z \quad & rot_z^{\text{pivot}} &= \gamma_{xy}
\end{aligned}
$$

On the faces normal to the X-axis, enforce

$$
\begin{aligned}
u_x(0, y, z) &= 0 \\
u_x(L_x, y, z) &= u_x^{\text{pivot}} L_x
\end{aligned}
$$

On the faces normal to the Y-axis, enforce

$$
\begin{aligned}
u_y(x, 0, z) &= 0 \\
u_y(x, L_y, z) &= u_y^{\text{pivot}} L_y
\end{aligned}
$$

On the faces normal to the Z-axis, enforce

$$
\begin{aligned}
u_z(x, y, 0) &= 0 \\
u_z(x, y, L_z) &= u_z^{\text{pivot}} L_z
\end{aligned}
$$

For the shear XY case, the boundary conditions are set as follows (ϵ_{xy} as specified by the user):

On faces normal to the X-axis, enforce

$$
\begin{aligned}
u_y(0, y, z) &= 0 \\
u_y(L_x, y, z) &= \gamma_{xy} L_x \\
u_z(0, y, z) &= 0 \\
u_z(L_x, y, z) &= 0
\end{aligned}
$$

On faces normal to the Y-axis, enforce

$$
\begin{aligned}
u_x(x, 0, z) &= 0 \\
u_x(x, L_y, z) &= 0 \\
u_z(x, 0, z) &= 0 \\
u_z(x, L_y, z) &= 0
\end{aligned}
$$

On faces normal to the Z-axis, enforce

$$u_z(x, y, 0) = 0$$
$$u_z(x, y, L_z) = 0$$

The boundary conditions for shear XZ can be obtained by switching the roles of y and z.

The boundary conditions for shear YZ can be obtained by switching the roles of x and y (starting from the shear XZ case).

Note that for the non-periodic shear test cases, we refrain from using additional degrees of freedom.

BETA

Release 2019 R3 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.