Rubber Material Curve Fitting with ABAQUS/CAE

2000 Michigan Regional ABAQUS Users' Meeting

Kurt Miller
kurt@axelproducts.com

Tod Dalrymple
tod@hks.com
Elastomers are Complex

Analytical Material Models do not Fully Describe Elastomers

Must Use Application Specific Experiments

ABAQUS/CAE & Training Can Help
Simple Tension
Simple Tension

\[ \lambda_3 = \lambda^{-1/2} \]

\[ \lambda_2 = \lambda^{-1/2} \]

\[ \lambda_1 = \lambda \]

10:1
Not this Simple!

Note: Stress and strain are total quantities.
Hyperelastic Material Models are More Complex

Stress (MPa) vs. Strain

 bèi äãñ
mì ěpüé-ê
pã éå-qéæãå
Pure Shear
Repeated Loading & Unloading

Stress (MPa) vs Strain

Initial Loading
Loading & Unloading to Varied Strain Levels

Axel Products, Inc.  www.axelproducts.com   734-994-8308
Loading Comparisons
Stress Relaxation

![Graph showing stress relaxation over time.](image-url)
Loading Comparison

![Graph showing stress-strain relationship for simple tension and various cycled conditions](graph.png)
What about Unloading?

![Graph showing engineering stress vs engineering strain with various cycling conditions and relaxed points at different times.](image-url)
Testing at Non-ambient Temperatures
Testing at Non-ambient Temperatures

![Graph showing engineering stress-strain relationship at different temperatures (-40C and 23C).]
Analytical Material Models do not Describe All Loading Conditions

Must Use Application Specific Loadings
Demo of Curve Fitting

- Starting ABAQUS/CAE
  - `abaqus cae`
  - This is the initial screen that pops up
  - Select the “Create Model Database” option
To define a hyperelastic material:

In the Module list located under the toolbar, click Property.

The cursor changes to an hourglass while the Property module loads.
The Property Module

- Notice the main menu bar has changed, and icons appear in the toolbox area.
- Choose the **Material Manager** icon
The Material Manager will appear. On the left side there is a list box of materials (the list is currently empty). The buttons on the right side are used to create and modify material definitions.

Select the Create button.

The Create Material dialog box appears.

In the Create Material dialog box a default material name Material-1 appears. Replace this default name by typing the name Rubber, and click Continue.
The Edit Material dialog
The material editor appears with a blank options list and option definition area.

From the menu bar in the upper portion of the editor window, select Mechanical->Elasticity->Hyperelastic.
The parameters and data corresponding to hyperelasticity appear in the option definition area below the option menus, and the word Hyperelastic appears in the Material Options list at the top of the window.

In the option definition area, accept Test data as the Input source selection. Note that the Strain energy potential button defaults to the value Unknown. Select the Suboptions button and select Uniaxial Test Data from the list that appears.
The Suboptions Editor appears.

Note that data required is Nominal (Engineering) Stress in column 1 and Nominal (Engineering) Strain in column 2. To display context-sensitive help for specific buttons, text fields, and other options in the Suboptions Editor, you must select the option of interest and then press [F1].

Click mouse button 3 in the first cell of the table, and select Read from File from the list that appears.
Finished reading in the experimental data

We are back in the Suboptions Editor but it may be rather small. Grab an edge or corner of the box and expand it until you can see all the Uniaxial test data values. There should be 14 data pairs, with maximum stress of 1.95 MPa and maximum strain value of 4.37 (437% strain).
Done with Test Data input

We are now back at the Edit Material dialog. If we wanted to bring in more test data, we repeat this process, selecting Biaxial Test Data, Planar Test Data, or Volumetric Test Data from the Suboptions pull-down menu. We are finished with test data input, so click OK in the Edit Material dialog.
Back to the Material Manager

- This puts us back to the Material Manager. Having read in the Uniaxial data we are ready to select the Evaluate function.
- The Evaluate option is useful in the following scenarios:

- Comparing test data with the behavior predicted by a particular strain energy potential.

- Evaluating multiple strain energy potentials.

- Viewing behavior predicted by coefficients for a particular strain energy potential.

- The default settings for Test Setup are shown below:
The defaults for **Strain Energy Potentials** are shown here. There are five major categories. For **Polynomial**, **Ogden**, and **Reduced Polynomial** you must select the number of terms, \( N \), in the energy function. You may choose one or more models to evaluate. In general selecting more than 1, 2 or 3 models makes viewing the results difficult.
Default Results Plotting of Evaluate Material

- The default results plotting uses 3 new viewports to display XY plots of stress-strain for Uniaxial response, Equibiaxial response and Planar Tension response. You may want to maximize each window in turn to better view the viewports.
Testing and Analysis of Elastomers

with ABAQUS

Analysis + Testing
Summary

- Elastomers are Complex
- Analytical Material Models do not Fully Describe Elastomers
- Must Use Application Specific Experiments
- ABAQUS/CAE & Training Can Help
Physical Testing Services for Engineering and Analysis

Training

ABAQUS Testing and Analysis of Elastomers with ABAQUS
October 3-5, 2000.

Technical Information
1. “Compression or Biaxial Extension?” (PDF: 304 KB / 3 pages).

Testing Services

Hyperelastic Properties of Elastomers

Experiments: Simple Tension, Pure Shear, Equal Biaxial, Compression & Volumetric

Data for: Ogden, Mooney-Rivlin, Arruda-Boyce, Damage Models, Viscelastic Decay

Elastic Plastic Properties of Plastics

Experiments: Modulus, Poisson’s Ratio

Data for: