Knowledge Base

Abaqus/CAE plug-in application for processing cyclic test data of filled elastomers and thermoplastics

Portfolio / Domain: SIMULIA Abaqus Unified FEA / SIMULIA Abaqus Unified FEA
Product: n/a

Information

QA Article: QA(
Last Update Date: 23.
Rating:
Views: 15(

MY FAVORITE CONTENT

QUESTION

I am modeling permanent set in a rubber-like material. How can I quickly process material test data in Abaqus/CAE to create the necessary modeling parameters?

ANSWER

(The following applies to Abaqus Versions 6.7-EF and higher.)

An Abaqus/CAE plug-in application for this purpose is attached below. It is meant to facilitate the computation of Abaqus material parameters from experimental data collected from cyclic loading tests of filled elastomers or thermoplastics. The plug-in allows you to examine and edit such test data and automatically create the necessary Abaqus material model options - including hyperelastic, Mullins effect, and plastic property specifications.

Note that it is not required that the test data contain permanent set or Mullins effect; that is, the plug-in may be used for a purely hyperelastic material. Further, it is not required that the data should be from a cyclic loading test; monotonic loading data can be processed as well.

Test Data Requirements

Figure 1 shows the typical experimental behavior of a filled elastomer when cyclically loaded between a given strain level and zero stress. Three distinct strain levels are shown (red, blue and purple curves) along with the primary loading path shown by the dashed line:

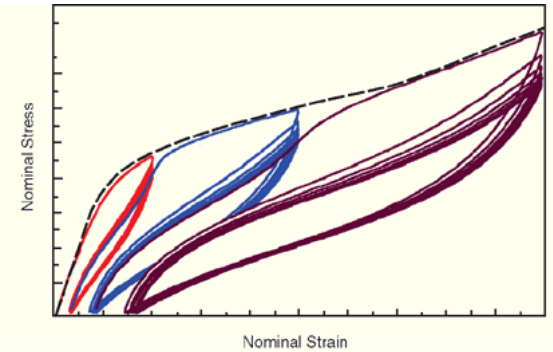

Figure 1: Typical cyclic loading behavior of a filled elastomer

The plug-in assumes that your cyclic (unloading / reloading) data is specified from a given unloading strain level to zero stress; many sets of cyclic data may be specified at different unloading strains but always cycling between a given strain level and zero stress. Therefore, if you specify cyclic data from a constant stress level, the resulting material may not behave as you intended. Additionally, in this case, the plug-in will assume that the stress does not increase along the primary loading path, and may result in unstable material.

Installation

To install the plug-in, save the attached archive file to one of the following directories:

abaqus_dir\abaqus_plugins where *abaqus_dir* is the Abaqus parent directory

home_dir\abaqus_plugins where *home_dir* is your home directory

current_dir\abaqus_plugins where *current_dir* is the current directory

Note that if the abaqus_plugins directory does not exist in the desired path, it must be created. The *plugin_dir* directory can also be used, where *plugin_dir* is a directory specified in the abaqus_v6.env file by the environment variable **plugin_central_dir**. You can store plug-ins in a central location that can be accessed by all users at your site if the directory to which **plugin_central_dir** refers is mounted on a file system that all users can access. For example,

plugin_central_dir = r'\\fileServer\sharedDirectory'

On Windows platforms, right click on the archive file and select **WinZip → Extract to here**. On Linux platforms, type **unzip plugin_1_2_1.zip** at the command prompt. A folder named abq_Fefp and a file named fefp_plugin.py will be extracted. Note that the plug-in will not function properly if this procedure is not followed.

Several sample files are also available for download:

- NR_uniaxial.csv, NR_biaxial.csv and st_nr.inp - the test data files are for a filled natural rubber ; the Abaqus input file is for simulation of simple tension that uses the material data output of the plug-in.
- Epdm_uniaxial.csv, st_epdm.inp - the test data file is for EPDM polymer ; the Abaqus input file is for simulation of simple tension that uses the material data output of the plug-in.
- 2007_ECCMR_PermanentSet.pdf - this paper illustrates simulation of permanent set in filled rubbers.

Usage

A menu item named **Create Rubber-like Material** will be available in the **Plug-ins** pull down menu from the Property or Visualization modules. Select **Plug-ins → Create Rubber-like Material...** as shown below:

*Figure 2: **Create Rubber-like Material...** menu*

Launching the plug-in will initiate a three step process to create the material model in Abaqus/CAE:

1. **Specify the test data**

   The first of three **Create Rubber-like Material** dialog boxes will be shown, as in Figure 3. You will be prompted to specify two comma-separated (.csv) files; one containing uniaxial test data and one containing biaxial test data. It is not required to specify both files, but you must select at least one. Note that the data file should contain nominal strain and nominal stress data pairs, with strain in the first column and stress in the second column. In the **Extract Mullins from:** drop-down menu, select which cycle the Mullins data should be extracted from for each unloading strain level.

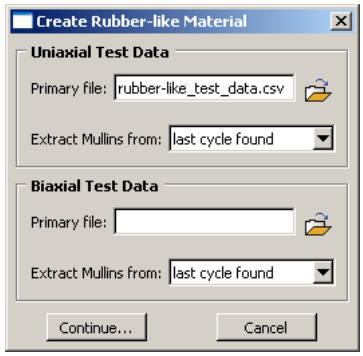

*Figure 3: First **Create Rubber-like Material** dialog*

2. **Extract, view, and edit the test data**

   The second **Create Rubber-like Material** dialog (Figure 4) allows you to extract, view, and edit the data. The dialog is divided into three panels: **Test Data Type**, **Curves to Plot**, and **XY Data**.

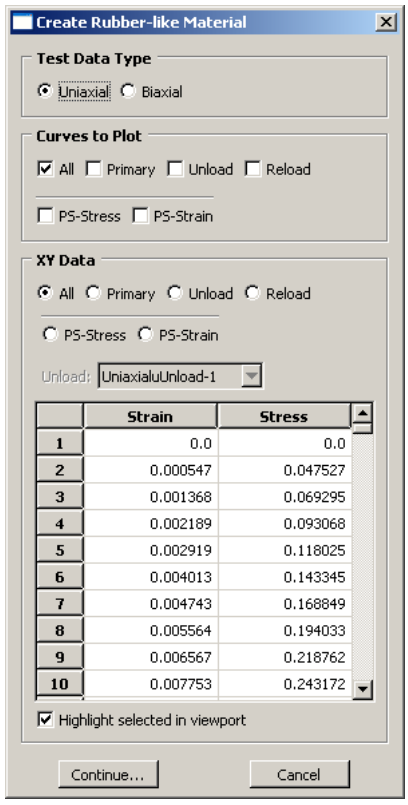

*Figure 4: Second **Create Rubber-like Material** dialog*

Test Data Type

If you selected only one test data type in the first dialog, you will not see the **Test Data Type** group box. If both data types were selected, use the radio button to view and edit the desired data type. No averaging or correlation is done in this step.

Curves to Plot

Select the desired curves to be plotted in the current viewport:

- **All** refers to all the raw data found in the selected test data file
- **Primary** refers to the primary loading curve, extracted by the plug-in and schematically shown as the dashed line in Figure 1.
- **Unload** refers to the unloading curve that will be extracted for each cyclic strain level
- **Reload** refers to the reloading curve that will be extracted for each cyclic strain level

  Note: The unloading and reloading curves will correspond to the cycle chosen in the **Extract Mullins from:** selection in Figure 3

If plastic data was found during the extraction step, two additional check boxes will be available. **PS-Stress** and **PS-Strain** refer to **P**ermanent **S**et stress (and strain) over permanent strain.

XY Data

In this area, you can control which plot data to edit. The same labels are found as above in the **Curves to Plot** area, but only one data can be edited at a time. The extracted data may be edited in this plug-in or in the **Visualization** module. Data types **All** and **Primary** have a right-mouse button menu named **Use as Initial Yield Point**. By selecting a cell or row in the table then selecting this menu, the stress value is placed as the first stress value in the **PS-Stress** data.

If the **Highlight selected in viewport** is checked, which is the default, any cell, row, or rows selected in the table area of the plug-in will be highlighted in the viewport.

This plug-in may also be used in elastic mode (no plasticity). If your test data only contains primary loading data, then, when entered into the plug-in, only some menus will be available, as shown in Figure 5.

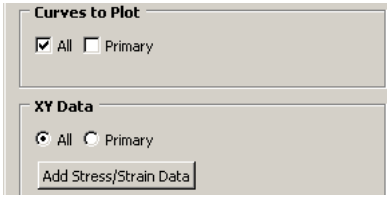

Figure 5: No plastic data menus

You may still enter plastic data by clicking on the **Add Stress/Strain Data** button. This will enable the **PS-Stress** and **PS-Strain** buttons. You may manually enter data in the table or use the right-mouse button menu **Read from File** to enter a .txt or .csv data file.

3. **Create the material model**

In the third **Create Rubber-like dialog**, you will be prompted to specify a material name. A material description may also be entered. If Mullins data has been entered in the previous dialog, the **Mullins Test Data** group box will be available. This will be for either uniaxial and/or biaxial loading. Select each level and cycle of the test data that you would like to include in the material to be created. Selecting (or de-selecting) check boxes will automatically plot the curves in the viewport.

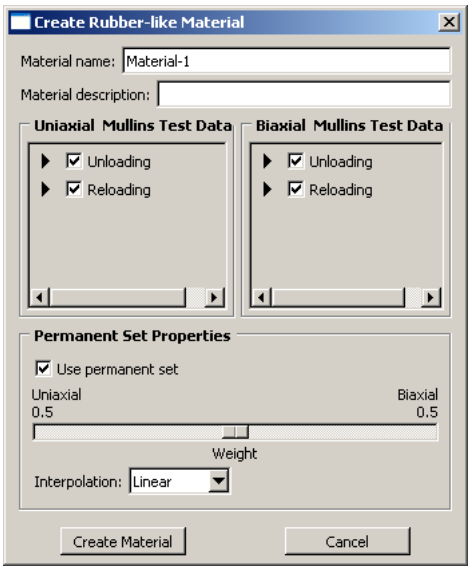

*Figure 6: Third **Create Rubber-like Material** dialog*

If plastic data has been entered in the previous dialog, the **Permanent Set Properties** area will be available. You can turn off the usage of plastic data by un-selecting the **Use permanent set** check box.

If both uniaxial and biaxial data have been entered in the previous dialog, additional control will be available for creating single isotropic hardening from the specified uniaxial and biaxial data. Move the weight bar to control the weight factors of uniaxial and biaxial hardening curve. Select an interpolation scheme between unloading curves from linear (default) or logarithmic interpolation.

The material model created through the plug-in will be available in the **Property** module.

Usage of **Permanent Set Properties** will create a material by converting the edited test data (including primary loading, any unloading / reloading, and permanent set data) to the following Abaqus material model options:

- *Hyperelastic test data corresponding to large strain elasticity, expressed in the stress-free configuration
- *Mullins effect test data, representing unloading / reloading behavior, expressed in the stress-free configuration
- *Plastic data, consisting of a single isotropic hardening curve expressed in terms of Kirchoff stress and equivalent plastic strain

For additional information see:

- Understanding the intermediate configuration when partitioning test data into *HYPERELASTIC and *PLASTIC subsets

Revision History

19 Dec 07	Version 1.1-1 of plug-in released.
05 Feb 08	Version 1.1-2 released to fix incorrect swap of stress/strain when there is no plastic data. Decreased filtering of noise data.
06 Mar 08	Version 1.1-3 released to fix last primary data point and final two data points extraction.
03 May 08	Version 1.1-4 released to fix extra checks on previous xy data objects.

04 Nov 08	Version 1.1-5 released to fix upper limit range error when finding first plasticity point.
16 Mar 09	Version 1.1-6 released to remove duplicate consecutive stress values from raw data.
05 May 09	Version 1.1-7 released to fix uniaxial non-permanent set additional data bug. For 6.9 release.
31 Aug 10	Version 1.2-1 for 6.10 release.
18 Apr 14	Version 1.3-1 for 6.14 release

KEYWORDS **test, data, edit, process, convert, plug, plug-in, fefp, plastic, hyper, elastic, material, plot, ru**

ATTACHMENT

st_epdm.inp | EPDM_uniaxial.csv | st_nr.inp | Answer_3522_Fig05.png | Answer_3522_Fig5a.png | plugin_1_1_7.zip | 2007_ECCMR_PermanentSet.pdf | NR_biaxial.csv | Answer_3522_Fig3a.png | plugin_1_2_1.zip | Answer_3522_Fig1a.png | NR_uniaxial.csv | Answer_3522_Fig2a.png | Answer_3522_Fig4.png | plugin_1_3_1.zip

SUBSCRIBE TO CHANGES ☐

RATING **On a scale of 1-5, how would you rate the technical content of the article?**

Please rate this article...

LET US KNOW IF THIS ARTICLE NEEDS TO BE ENHANCED

UNCLEAR | MISSING INFO | DUPLICATE | OUT OF DATE | ERROR DETECTED

See Comments (1)