

Knowledge Base

Information



Plug-in utility to convert nominal plastic stress-strain data

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

QA Article: QA00000009204e
Last Update Date: 24.11.2020
Rating: 5.0
Views: 996

QUESTION

Abaqus requires that elastic-plastic material definitions be entered in terms of elastic modulus and true stress - logarithmic plastic strain. Are there any automated ways to convert my nominal stress-strain data to the true stress - logarithmic plastic strain format that Abaqus requires?

ANSWER

(The attached plug-in applies to Versions 6.6-1 and higher; the attached GUI custom application applies to Versions 6.5-1 and higher.)

The Abaqus Analysis User's Manual presents the equations necessary to convert nominal stress-strain data of an isotropic material to true stress - logarithmic plastic strain data. Abaqus does not have a built in feature to perform this conversion. However, Abaqus does include the means to allow users to extend its "off-the-shelf" capabilities.

A utility that allows you to:

- plot and manipulate raw nominal stress-strain test data,
- determine an elastic modulus and proportional limit from it, and
- write it out in the true stress - logarithmic plastic strain format

is attached below for download. It is available as a plug-in or a custom application.

Installation

Plug-in version

To install the plug-in, save the files in the attached archive nom2true_plugin.zip 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'
```

The next time Abaqus/CAE is started, a menu item named **Nominal-True...** will be available in the **Plug-ins** pull down menu in the **Property** module.

Custom Application Version

The attached archive nom2true_customApp.zip contains the files necessary to install and run the Abaqus/CAE add-on. To install the add-on and make it available from any directory, perform the following tasks.

1. Determine the location of your Abaqus installation using the command

```
abaqus whereami
```

For the Windows platform the location is usually `C:\ABAQUS\6.n-1`.

2. Create a new subdirectory in the Abaqus installation directory called `customApps\nom2true`. For the Windows platform the entire path will then be, for example, `C:\ABAQUS\6.4-1\customApps\nom2true`.
3. Copy the attached Python files to the new `nom2true` subdirectory.
4. Within the site subdirectory of your installation, edit the file `abaqus.app` and add the following line:

```
mycae cae -custom myCaeApp
```

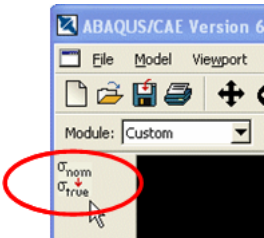
5. In the same site subdirectory, edit the file `abaqus.aev` so that it includes the path to the new custom files. For example, on the Windows platform you should change from this:

```
PYTHONPATH
$ABA_HOME/cae/Python/Lib:$ABA_HOME/cae/Python/Obj:
$ABA_HOME/cae/exec/lbr:.$PYTHONPATH
```

to this:

```
PYTHONPATH
$ABA_HOME/cae/Python/Lib:$ABA_HOME/cae/Python/Obj:
$ABA_HOME/cae/exec/lbr:$ABA_PATH/customApps/nom2true:
.$PYTHONPATH
```

If the installation was successful, you will now be able to use the Abaqus/CAE customization. To do so, start Abaqus/CAE with the command `abaqus mycae`. A new module, named **Custom**, will be shown in addition to the given Abaqus/CAE modules. The **Custom** module includes a single new menu item and a single new toolbox button:



MY FAVORITE CONTENT

Usage

When the plug-in or custom application is launched, the following dialog will appear:

Nominal Stress-Strain Processing

Raw Data File

File name:

Read nominal strain values from column:

Read nominal stress values from column:

Field delimiter: spaces, tabs or commas

Comment lines beginning with "***" are allowed.

Data Filter

☒ Read all rows

☐ Skip rows between reads

Manipulate Raw Data

☐ Shift all points by (dX,dY):

☐ Scale all points by (X,Y):

☐ Extrapolate to max. strain of:

☐ Allow decreasing stress values

Elastic Modulus & Proportional Limit

☒ Specify elastic modulus and proportional limit

E: Prop. Limit:

☐ Auto estimate elastic modulus and proportional limit

Goodness of fit, R-squared, for elastic modulus:

Save Options

☐ Save to file:

☐ Save to model:

Material name:

Poisson's ratio:

The tool has several features that can be used to generate the desired elastic-plastic material definition; the specifics of each feature are defined in the dialog box as follows:

Raw Data File

The file containing the raw nominal stress-strain data to be processed is specified here. The application expects the data to be in column format, and allows the user to select the appropriate column for stress and strain. Note that the data can be delimited by spaces, tabs, or commas. The user may add comments to the data file by including ** at the beginning of any comment line.

Data Filter

A simple filtering of the data can be achieved by specifying the desired number of data pairs to be skipped between points. Skipping rows may be useful in cases where the data acquisition instrumentation recorded numerous samples per second.

Manipulate Raw Data

Raw data from testing seldom starts at the (0,0) point needed for analysis purposes. Also, raw data may be given in a different unit system than desired, or in terms of percent strain. This portion of the dialog box allows a shift and scale of the data as desired.

Often times the stress-strain data does not extend as far as an analyst would like. A capability to extrapolate the data to a given strain is included. The data is extrapolated by simply finding the slope of the last two processed data pairs and using it to calculate the stress at the desired extrapolated strain. The strain specified by the user is nominal strain.

Abaqus requires that the strain defined in an elastic-plastic material definition be always increasing. The application checks the raw data and automatically skips any data pairs which have less strain than the preceding data pair. In addition, the application allows the user to specify whether or not decreasing stress values (negative slope) are allowed. If decreasing stress values are not allowed (the default), any data pair with a stress value less than that of the preceding data pair is skipped.

Note: Strain is incorrectly scaled when the strain scale factor is less than one and when the "Allow decreasing stress values" box is not checked. To work around the problem, either place a check in this box or scale the strain before using the application.

Elastic Modulus & Proportional Limit

Two mutually exclusive methods of determining the elastic modulus and proportional limit of the nominal stress-strain curve are provided.

The first method allows the user to directly specify the elastic modulus and proportional limit. Once these two parameters are entered, the user can press the **Plot** button to plot the stress-strain data against the given elastic modulus and proportional limit. If the elastic modulus and proportional limit entered do not match the stress-strain curve well, the user can modify the elastic modulus and proportional limit easily and press the **Plot** button again, doing this iteratively until a good match is found.

The second method automatically determines an elastic modulus and proportional limit based on a linear least-squares curve fit. A "goodness of fit" known as "R-squared" must be requested through a slider bar in the dialog box. It is a measure of how well the linear curve fit matches the given data. The value can vary between zero and one, with one being a perfect fit. In the auto estimation method, the user first enters the desired "R-squared" value. When the **Plot** button is pressed, the application starts with the first two data pairs and calculates "R-squared." Then the next data pair is added to the previous two and "R-squared" is calculated again. This is done repeatedly, until the value of "R-squared" becomes less than that specified by the user. At this point the proportional limit is defined and the slope of the best fit line is calculated, providing an estimate of the elastic modulus.

In cases where there is significant deviation in the test data, the "R-squared" method of estimating the elastic modulus and proportional limit will not work well. This is because significant deviation causes the "R-squared" value to become low after a relatively few number of data pairs, which will result in an unrealistically low estimate of the proportional limit and possibly an inaccurate estimate of the elastic modulus. To counter this, the user can increase the number of points being skipped or decrease the value of "R-squared." However, in some cases it will be better for the user to specify the elastic modulus and proportional limit manually and iteratively than have it performed automatically.

Save Options

Two save options are available, saving the material input data to a file or copying the material definition to any of the current models in the .cae database. When saving to a file, an input file snippet with the appropriate keywords (*ELASTIC and *PLASTIC) will be written to disk. The input file snippet will be written in the form required by Abaqus, namely true stress and logarithmic plastic strain. The input file snippet can be copied directly into any input file. When copying to a database, the target model can be selected from a combo box that is populated with all of the current models. A name must be specified for the new material. If the new material name matches one already defined in the selected model an appropriate warning message is posted.

Additional Notes

It should be noted that the graphs of raw data, processed data, and elastic-plastic data that are drawn when the user presses the **Plot** button are saved during the session and can be manipulated. To do so, the user must first switch to the **Visualization** module. From there, all the XY plotting tools and options can be used on the curves. The **XY Data Manager** is available under the **Tools** → **XY Data** → **Manager...** menu item. The XY curves and XY plot can be modified using the **Options** menu item. Finally, the curves can be probed by selecting the **Tools** → **Query...** menu item. These options and tools are useful for zooming into the region of the curve near the proportional limit so that the proportional limit and elastic modulus can be selected accurately.

For additional information see:

- 'Inelastic behavior,' Section 23.1.1 of the Abaqus 6.12 or 6.13 Analysis User's Guide

Revision History

28 Jul 10	Release 1.2-4
-----------	---------------

Disclaimer

The attachments to this article are subject to certain usage conditions. Please [click here](#) for details.

KEYWORDS plasticity, true stress, true strain, convert, conversion, nominal, logarithmic, plastic, inelastic,

ATTACHMENT	Answer_1739_Fig1a.png	Answer_1739_Fig2.png	nom2true_plugin.zip	nom2true_customApp.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