

# Knowledge Base

## Information



## Abaqus/CAE plug-in utility to calibrate Nitinol material behavior

**Portfolio / Domain:** SIMULIA Abaqus Unified FEA / SIMULIA Abaqus Unified FEA  
**Product:** SIMULIA Abaqus/CAE

**QA Article:** QA00000020466e  
**Applicable Level:** 6.10-0  
**Last Update Date:** 09/10/2020  
**Rating:** 5.0  
**Views:** 104

### QUESTION How can I calibrate the Nitinol material model using test data?

**ANSWER** (The following applies to Abaqus 6.10-EF and later releases.)

This answer provides an Abaqus/CAE plug-in for calibrating the superelastic and/or the superelastic-plastic behaviors of Nitinol for Abaqus/Standard or Abaqus/Explicit analyses. Please note that the stress/strain measures used are true stress and log strains.

The plug-in provides a user friendly interface to create Nitinol material parameters based on uniaxial tension test data. Necessary keywords and datalines will be generated automatically based on a few characteristic points selected from the test data. The material will be renamed according to the required naming convention for Nitinol and can be directly used inside Abaqus/CAE.

In addition the plug-in offers an option to evaluate the material definition with a uniaxial 3D one-element model. If you select this option, the plug-in submits the analysis and plots the results with the original test data for comparison. You can iterate the above process by editing the points picked on the test data until the desired material behavior is achieved.

#### 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'`

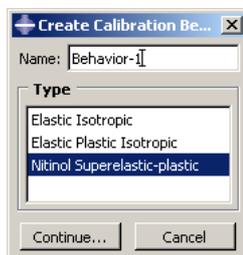
To use Nitinol superelastic-plastic material behavior, on Windows platforms, right click on the archive file and select **WinZip** → **Extract to here**. On Linux platforms, type **unzip nitinolSuperElastPlast.zip** at the command prompt. A folder named nitinolSuperElastPlast will be extracted.

Note that the plug-in will not function properly if this procedure is not followed.

#### Usage

The plug-in can be executed from the **Property** module only. See Abaqus Answer Creating custom material calibration plug-ins in Abaqus/CAE for more details on running calibration behavior plug-ins.

You can either create or import the Nitinol stress/strain test data and plot the curve in Abaqus/CAE. When creating a Nitinol material behavior you must select **Nitinol Superelastic-plastic** in the **Create Calibration Behavior** dialog.

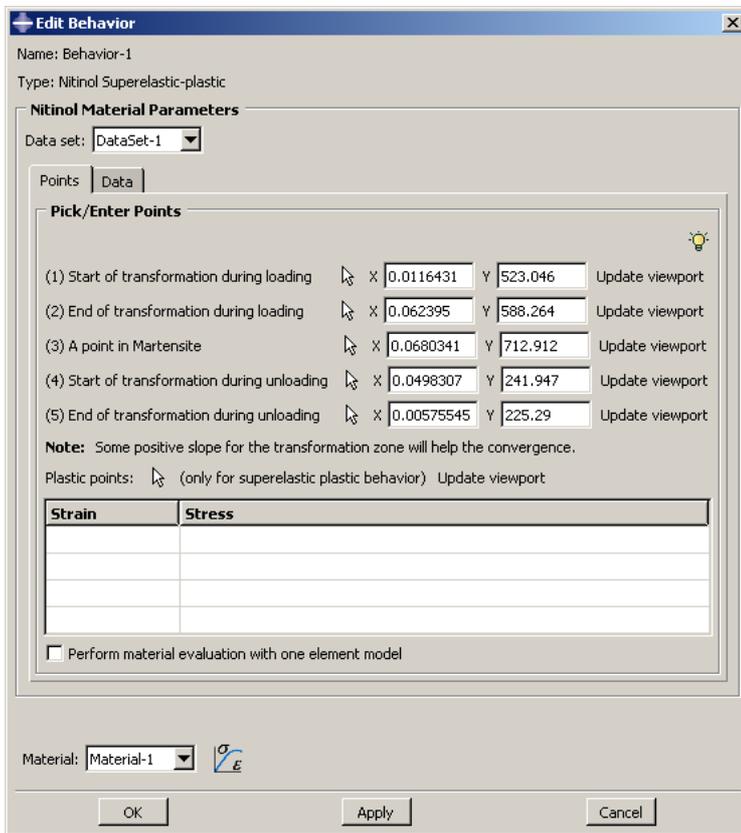


Select **Continue** to invoke the **Edit Behavior** dialog. In the **Edit Behavior** dialog select the desired data set under the **Data set** pulldown. You will need to have the data set active in the viewport in order to select points. Complete the material definition by filling out the **Points** and **Data** tabs.

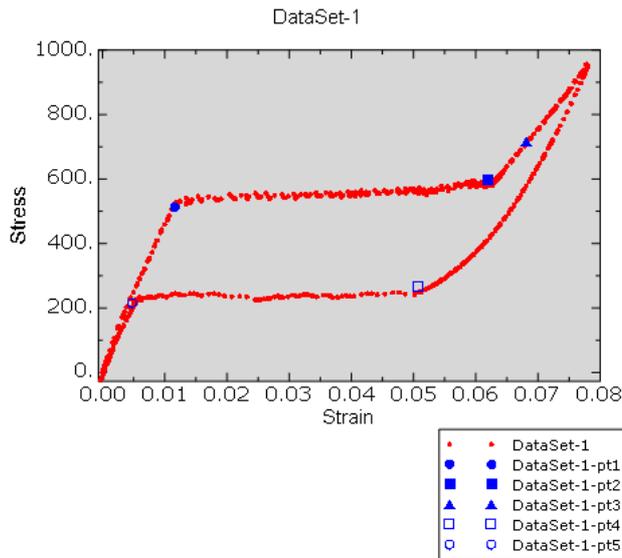
#### Points tab

You must pick the five characteristic points on the stress-strain plot using the **Pick** buttons for each point. When a point is picked the stress and strain values for the respective point populate the corresponding X and Y text fields. The stress-strain values for the picked points are used to calculate the Nitinol material constants. There is a tip button (available on the right-side of the dialog) that assists you regarding the approximate locations of the points on the plot. Be careful when picking the points, since having a positive slope between points 1 and 2, and between points 4 and 5 help the analysis converge.

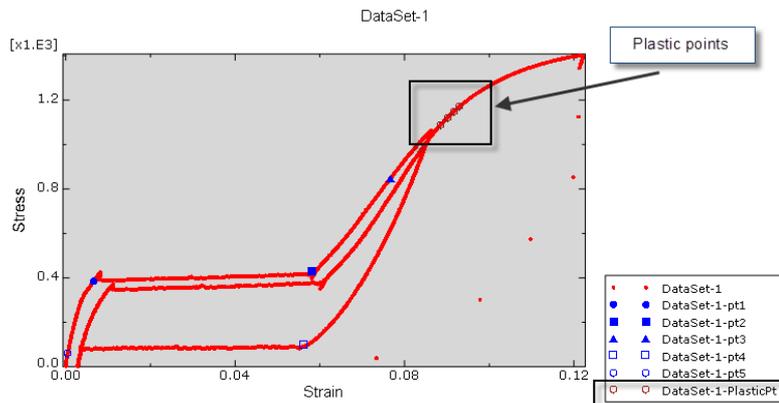
You can also change the stress-strain point values from the plug-in dialog by directly editing the text fields. In order to have visual feedback of the change made in the text fields, you must click the **Update viewport** button. This will update the point locations and display the latest locations on the plot.



The image below shows the picked points on the stress-strain curve for a superelastic material behavior.



When superelastic-plastic behavior is to be calibrated from the test data, you must pick multiple points from the plotted data. Select the **Plastic points** white arrow to begin selecting points.



The points with corresponding stress and strain values are populated in the plug-in dialog box table. The plastic point picking procedure **must** be canceled from the prompt area once all the required plastic points are picked. Use the red cross shown below to cancel out of the picking procedure.



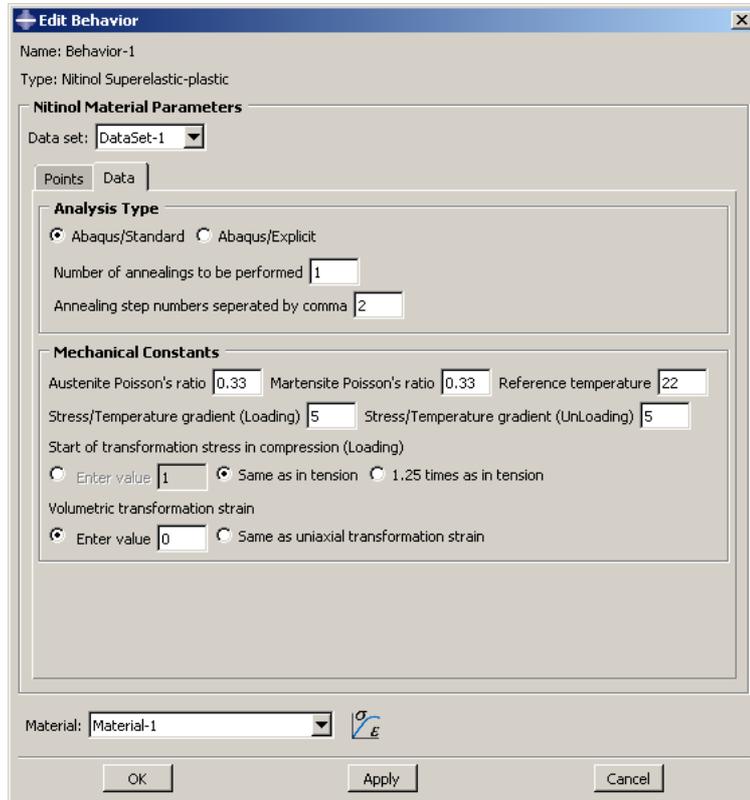
You can also manually add the stress-strain values of the plastic points. As described above, the **Update viewport** button updates the plot of the plastic points.

**Data tab**

Select the analysis type.

For Abaqus/Standard, you need to specify the number of annealings to be performed and the steps in which they will occur. When multiple steps are to be used, the entered step numbers must be comma separated.

For Abaqus/Explicit, you must specify if the usage will be for 1D, 2D or 3D elements and also specify the material density. Mechanical constants such as austenite Poisson's ratio, martensite Poisson's ratio, reference temperature, stress/temperature gradient during loading and unloading, start of transformation stress in compression and volumetric transformation strain must be entered.



The information entered as shown above is used to create or edit an existing material. There are material name requirements to use this tool.

For Abaqus/Standard, the name should start with **ABQ\_SUPER\_ELASTIC** and for Abaqus/Explicit the name should start with **ABQ\_SUPER\_ELASTIC\_N3D**, **ABQ\_SUPER\_ELASTIC\_N2D** or **ABQ\_SUPER\_ELASTIC\_N1D**.

If the user specified material name does not conform to the rule, the plug-in will add the required string to the name. For instance if you select Abaqus/Standard as the analysis type and specify the material name as 'Material-1', the plug-in changes the name to 'ABQ\_SUPER\_ELASTIC\_Material-1'. It is your responsibility to make sure that the material name follows the naming conventions of Abaqus/CAE. The plug-in writes the required \*USER MATERIAL and \*DEPVAR keywords and provides descriptions of the solution dependent variables (SDVs) under the \*DEPVAR keyword, which helps during post-processing of results. The image below shows a sample keyword editor image for the superelastic behavior. Similarly appropriate constants are written for superelastic-plastic behavior.

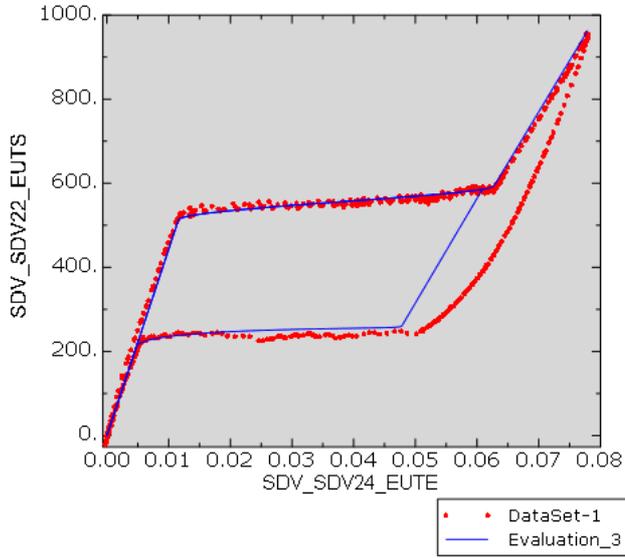
```

** MATERIALS
**
*Material, name=ABQ_SUPER_ELASTIC_Material-1
*Depvar
24,
1, SDV1_EE11, "Linear elastic strain component"
2, SDV2_EE22, "Linear elastic strain component"
3, SDV3_EE33, "Linear elastic strain component"
4, SDV4_EE12, "Linear elastic strain component"
5, SDV5_EE13, "Linear elastic strain component"
6, SDV6_EE23, "Linear elastic strain component"
7, SDV7_TE11, "Transformation strain component"
8, SDV8_TE22, "Transformation strain component"
9, SDV9_TE33, "Transformation strain component"
10, SDV10_TE12, "Transformation strain component"
11, SDV11_TE13, "Transformation strain component"
12, SDV12_TE23, "Transformation strain component"
19, SDV19_ETE, "Equivalent transformation strain"
20, SDV20_VTE, "Volumetric transformation strain"
21, SDV21_FM, "Fraction of martensite"
22, SDV22_EUTS, "Equivalent uniaxial tensile stress"
23, SDV23_EUTTE, "Equivalent uniaxial tensile transformation strain"
24, SDV24_EUTE, "Equivalent uniaxial tensile total strain"

*User Material, constants=16, unsymm
44217.5, 0.33, 24351.3, 0.33, 0.0383822, 5., 512.936, 593.384
22., 5., 260.301, 211.063, 512.936, 0., 1., 2.
    
```

The single element evaluation tool creates a model with the mapped material and required load and boundary conditions, submits a job for uniaxial tension analysis and does the post-processing.

For each mapped material a new one element model is created in Abaqus/CAE and a job is submitted. During post-processing the plug-in creates an X-Y plot of analysis result SDV22\_EUTS (Equivalent uniaxial tensile stress) and SDV24\_EUTE (Equivalent uniaxial tensile total strain) and plots it on the same viewport as the test data. For superelastic behavior a representative image of the X-Y plot is shown below.



You can iterate the entire process by re-picking the points or by entering different data in the plug-in dialog and running the one element model until the required material behavior is accomplished.

**Notes**

- Please note that the one-element-eval.odb is overwritten every time when the material behavior is edited and the one element analysis is performed.

**Disclaimer**

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

**KEYWORDS**     **plug-in, nitinol, stent**

**ATTACHMENT**

- answer\_4699\_fig4.png
- answer\_4699\_fig3.png
- answer\_4699\_fig2.png
- answer\_4699\_fig6.png
- nitinolSuperElastPlast.zip
- answer\_4699\_fig5a.png
- answer\_4699\_fig1.png
- answer\_4699\_fig5b.png
- answer\_4699\_fig7.png

**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