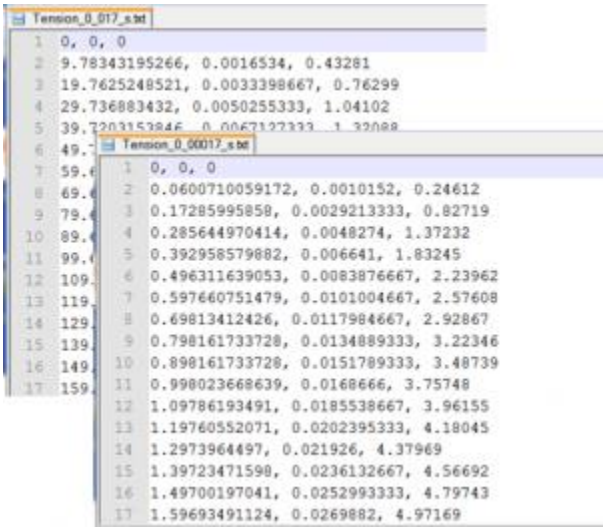
Introduction to MCalibration®

In this tutorial you will learn to:

- Read a set of experimental data into MCalibration
- Use that data to calibrate a material model
- Use virtual experiments to examine the behavior of a calibrated material model
- Export the material model to a finite element input file

Experimental Data



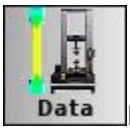
This simple example uses uniaxial tension data at two different strain rates. You may download the experimental data files here: TensionData1 and TensionData2. The figures to the right show parts of the data files using a text editor.

The following page contains more info about what experimental data MCalibration requires.

MCalibration Main Window

Before reading the experimental data let's examine the different parts of the main window. The main window has 4 different sections:

Welcome: This section can be used to open recently used calibration files (called mcal-files).



Data: This section is used to view and edit experimental data.

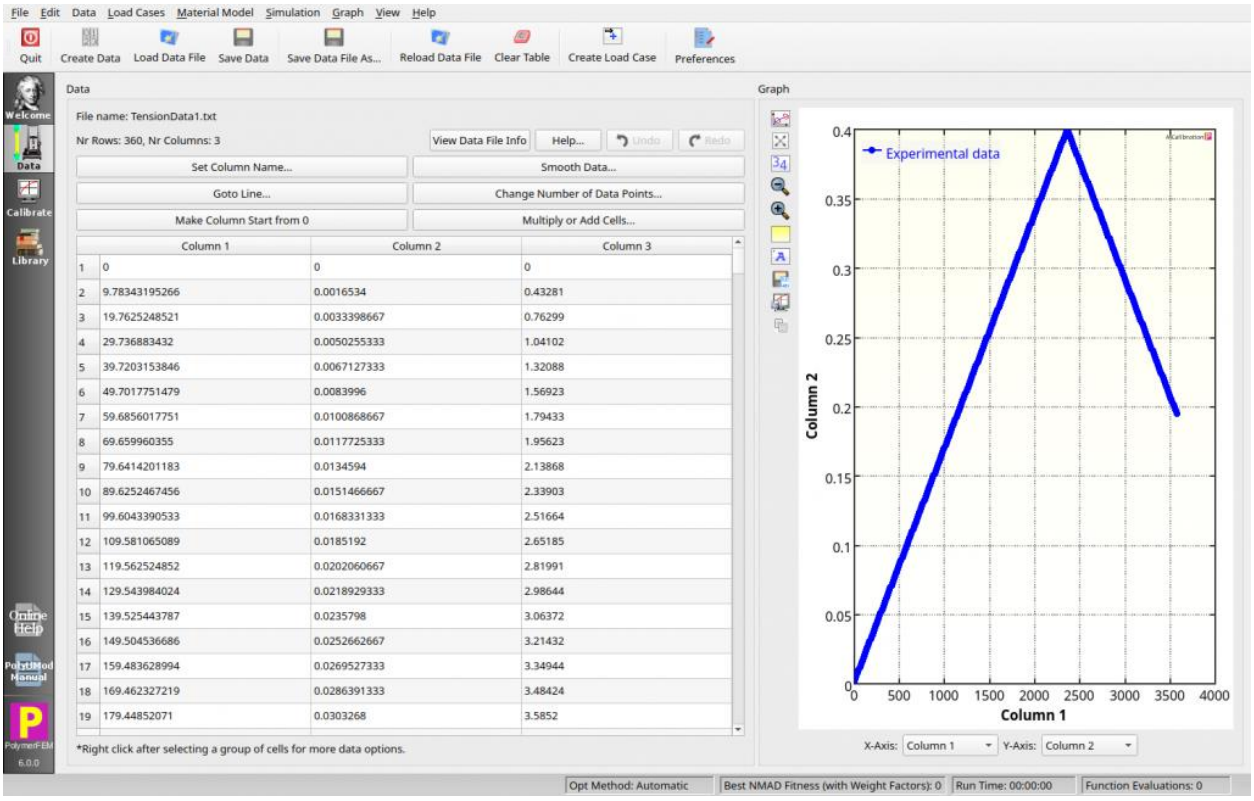
Calibrate: This section is used to calibrate material models, and to examine the response of a material model.

Library: Contains a user's collection of already calibrated models for different materials.

Data Section

Switch to the Data section by clicking on the Data icon in the toolbar to the left.

Click on Load Data File to read in the first experimental data downloaded above (TensionData1).

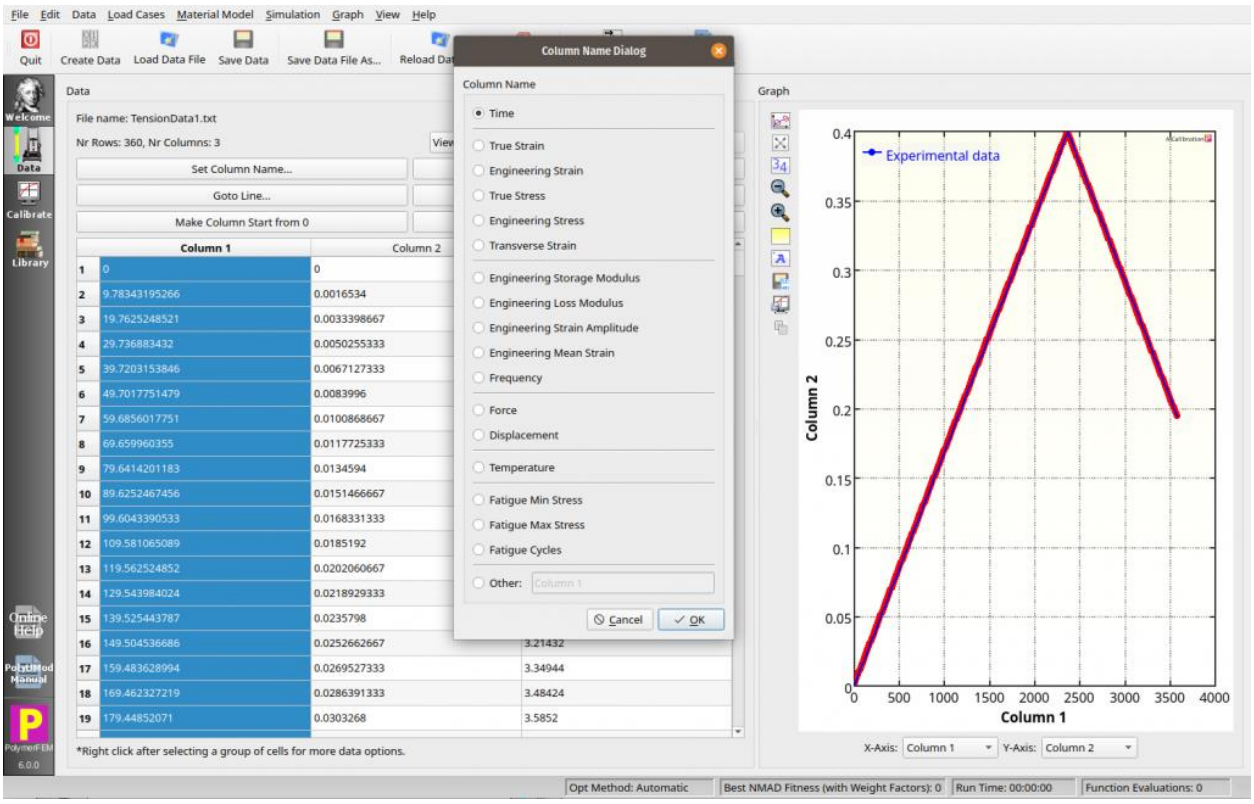


Next, we specify what the different columns of data contain.

1. Select Column 1 (or bring up a context menu by right-clicking in the column).
2. Then select Set Column Name.
3. Select Time and click OK.

This assigns column 1 as a time column.

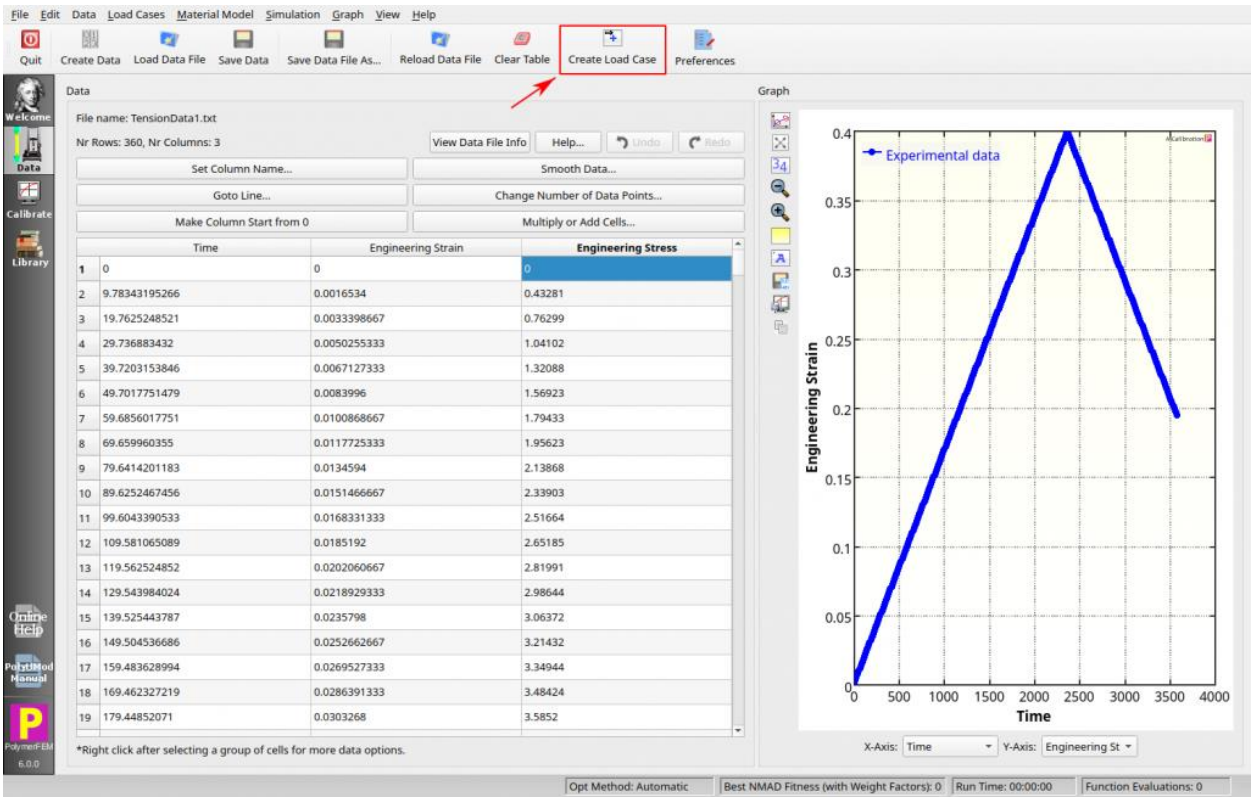
Repeat these steps to assign column 2 as engineering strain, and column 3 as engineering stress.



Start creating a “load case” for the material model calibration by clicking on the Create Load Case button.

Note: A load case is the same as an experimental test that can be used for material model calibration.

Note: The Data Section contains many functions for making experimental data suitable for material model calibration.

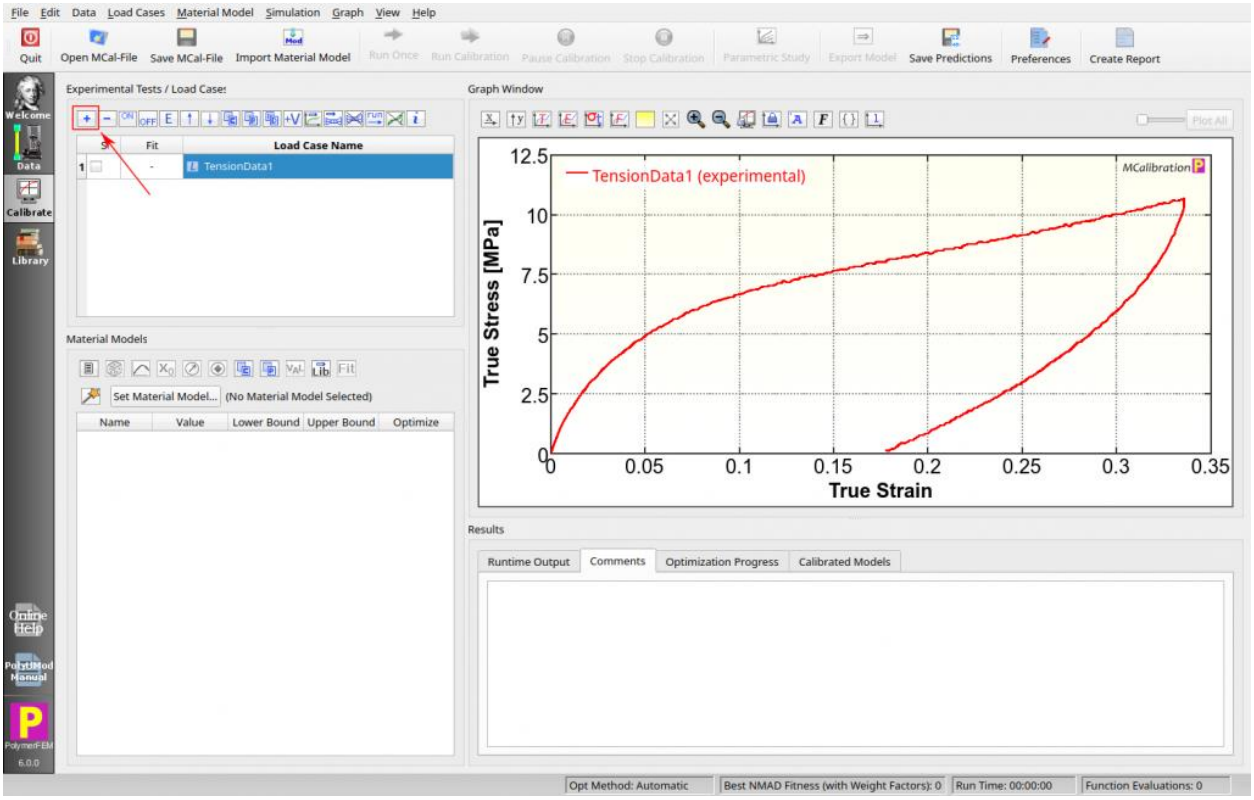

File Edit Data Load Cases Material Model Simulation Graph View Help
Quit
Create Data
Load Data File
Save Data
Save Data File As...
Reload Data File
Clear Table
Create Load Case
Preferences
Welcome
Data
Calibrate
Library
Online Help
PolyMod Manual
PolymerFEM 6.0.0
Data
File name: TensionData1.txt
Nr Rows: 360, Nr Columns: 3
View Data File Info
Help...
Undo
Redo
Set Column Name...
Smooth Data...
Goto Line...
Change Number of Data Points...
Make Column Start from 0
Multiply or Add Cells...
Time
Engineering Strain
Engineering Stress
1 0 0 0
2 9.78343195266 0.0016534 0.43281
3 19.7625248521 0.0033398667 0.76299
4 29.736883432 0.0050255333 1.04102
5 39.7203153846 0.0067127333 1.32088
6 49.7017751479 0.0083996 1.56923
7 59.6856017751 0.0100868667 1.79433
8 69.659960355 0.0117725333 1.95623
9 79.6414201183 0.0134594 2.13868
10 89.6252467456 0.0151466667 2.33903
11 99.6043390533 0.0168331333 2.51664
12 109.581065089 0.0185192 2.65185
13 119.562524852 0.0202060667 2.81991
14 129.543984024 0.0218929333 2.98644
15 139.525443787 0.0235798 3.06372
16 149.504536686 0.0252662667 3.21432
17 159.483628994 0.0269527333 3.34944
18 169.462327219 0.0286391333 3.48424
19 179.44852071 0.0303268 3.5852
*Right click after selecting a group of cells for more data options.
Graph
Experimental data
Engineering Strain
Time
0 0.05 0.1 0.15 0.2 0.25 0.3 0.35 0.4
0 500 1000 1500 2000 2500 3000 3500 4000
X-Axis: Time
Y-Axis: Engineering St
Opt Method: Automatic
Best NMAD Fitness (with Weight Factors): 0
Run Time: 00:00:00
Function Evaluations: 0

Calibrate Section

MCalibration switches to the Calibrate section when the save button is clicked in the load case dialog.

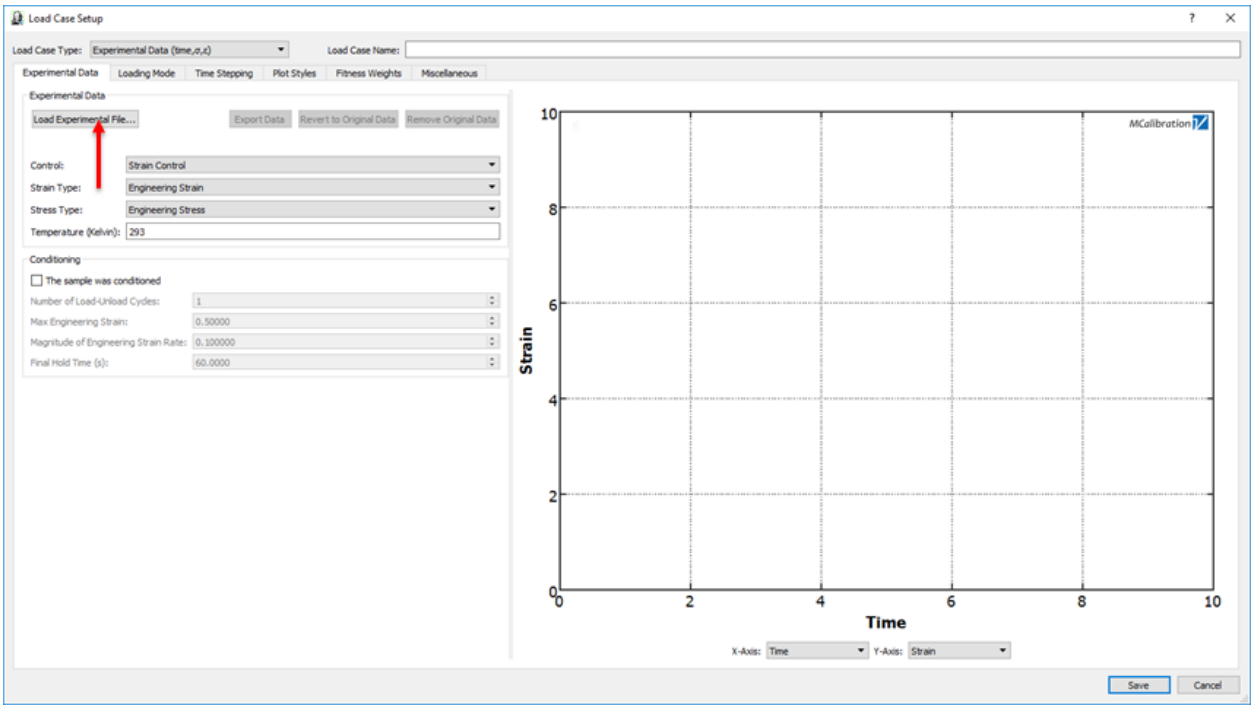
Before starting the calibration we need to read in the second experimental data file. To do this we can switch to the Data section and repeat the steps just performed. Instead, here we will illustrate another way.

- Click on the + button to add a Load Case.



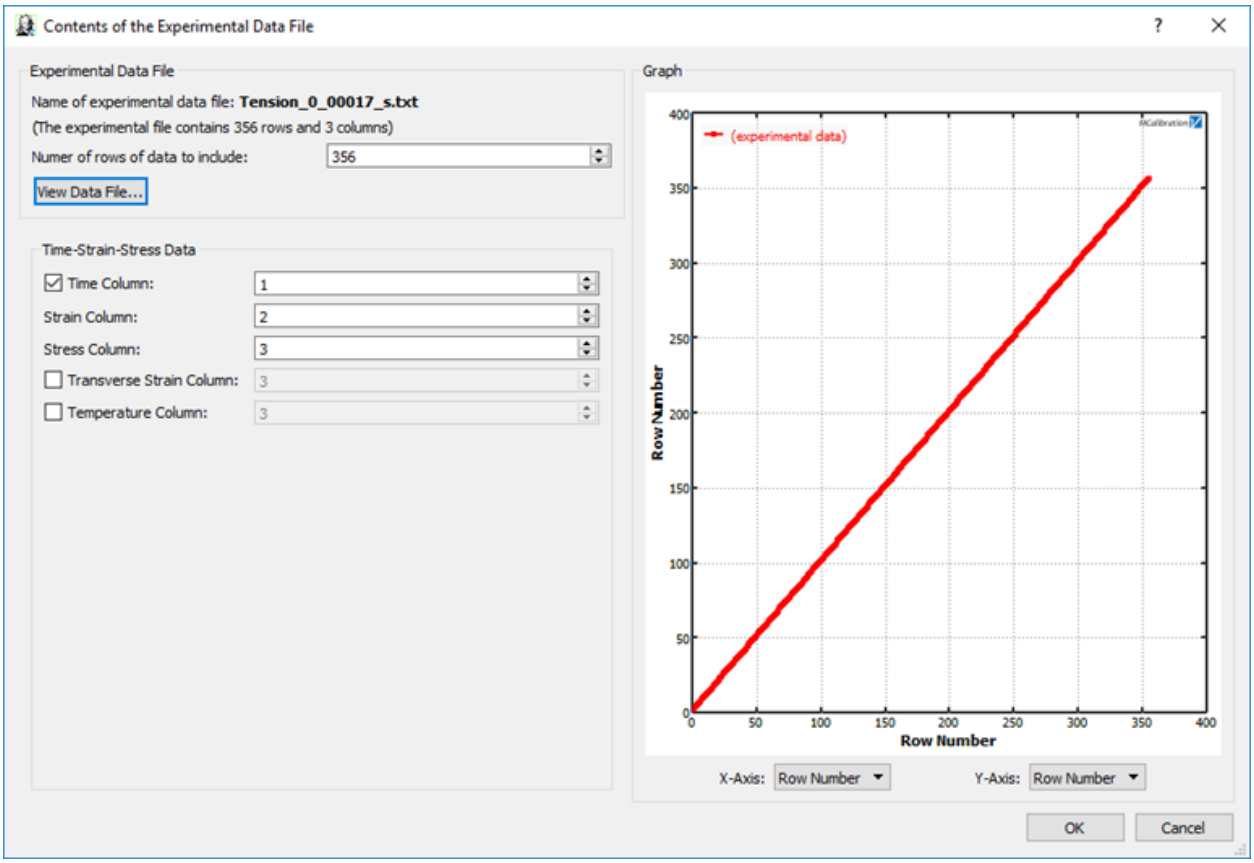
This brings up an empty Load Case dialog box.

- Click on the Load Experimental File... button.
- Select the TensionData2.txt file downloaded earlier. Then click OK.



This opens a dialog box that is used to specify the contents of the experimental data file

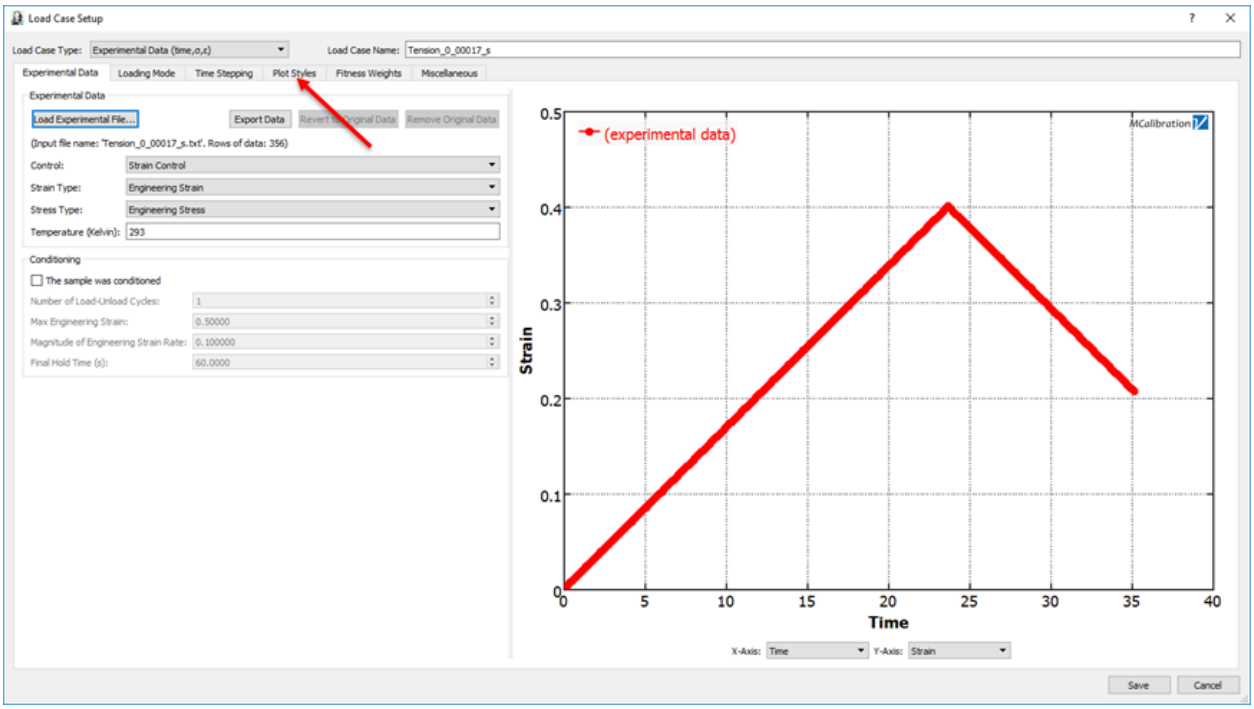
- Here time is in column 1, strain is in column 2, and stress is in column 3.
- Click the OK button.



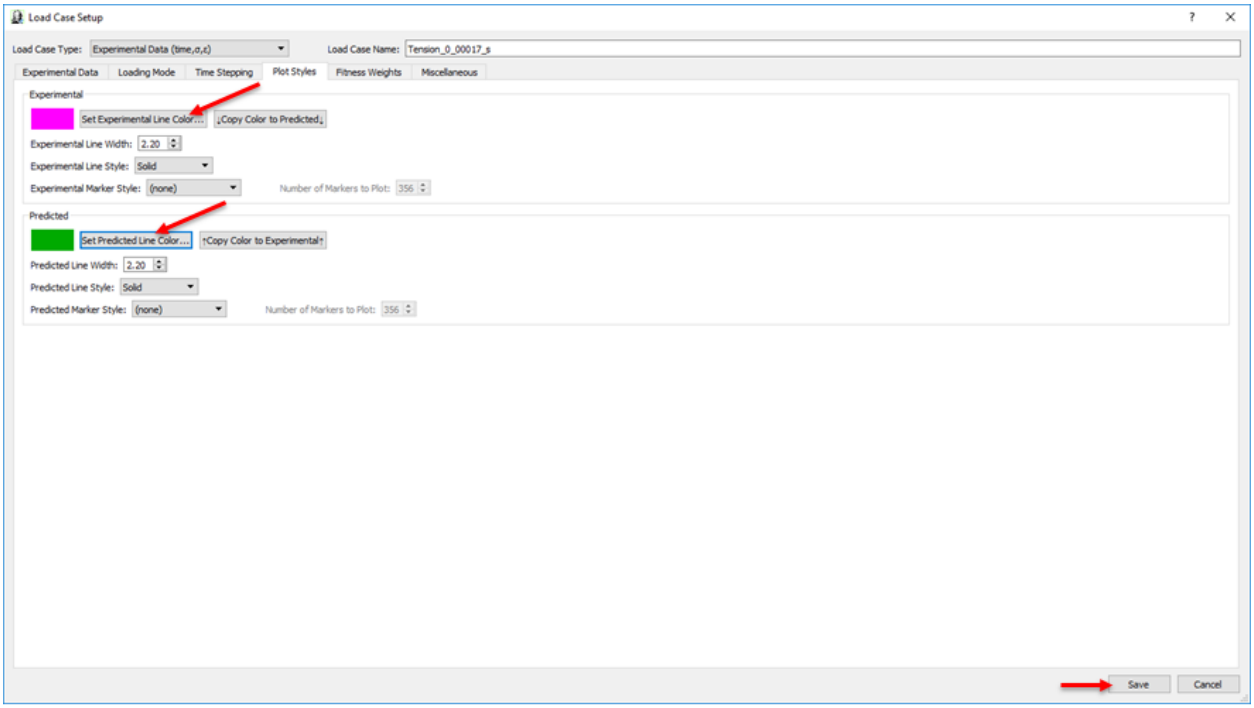
This loads the experimental data into the Load Case Setup dialog box.

Most of the default settings are OK, but let's change the line colors of this load case.

Click on the Plot Styles tab.



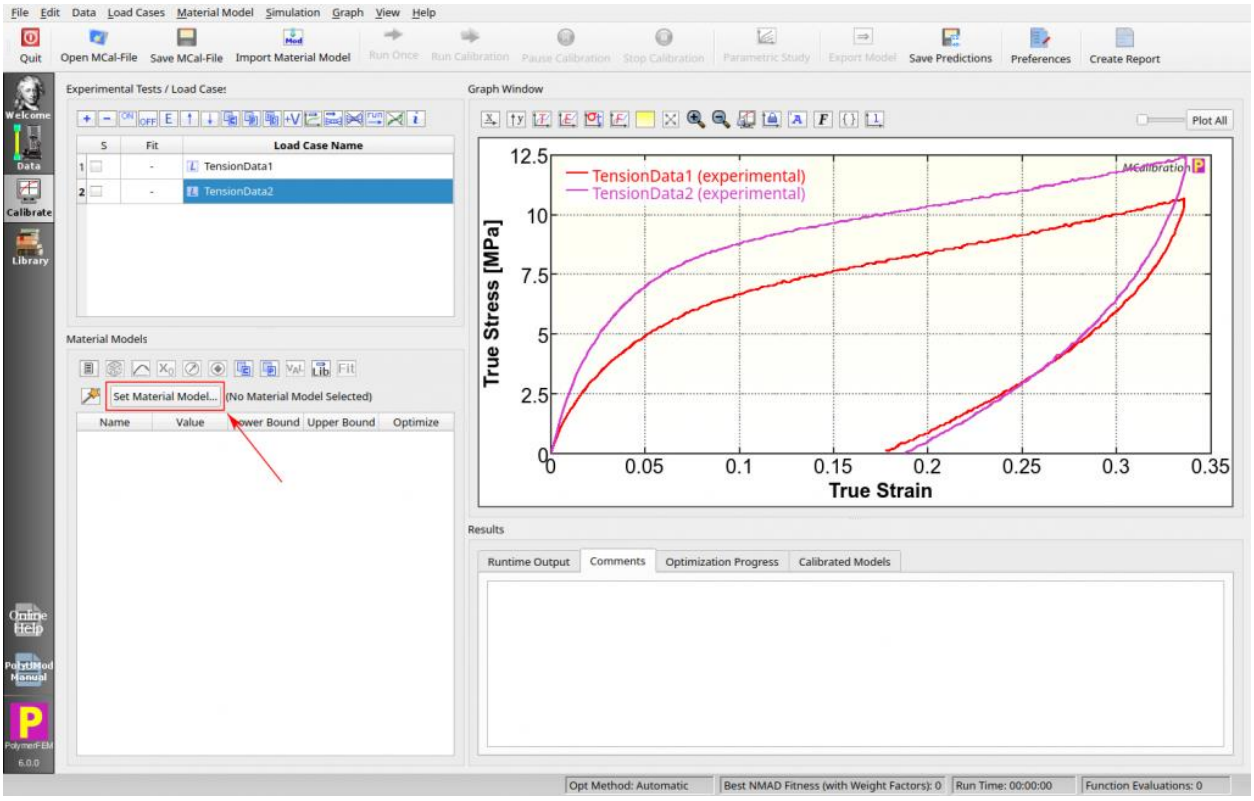
- Click on the
  Set Experimental Line Color... button.
- Click on the Set Predicted Line Color... button
- Click Save when done.



Material Model

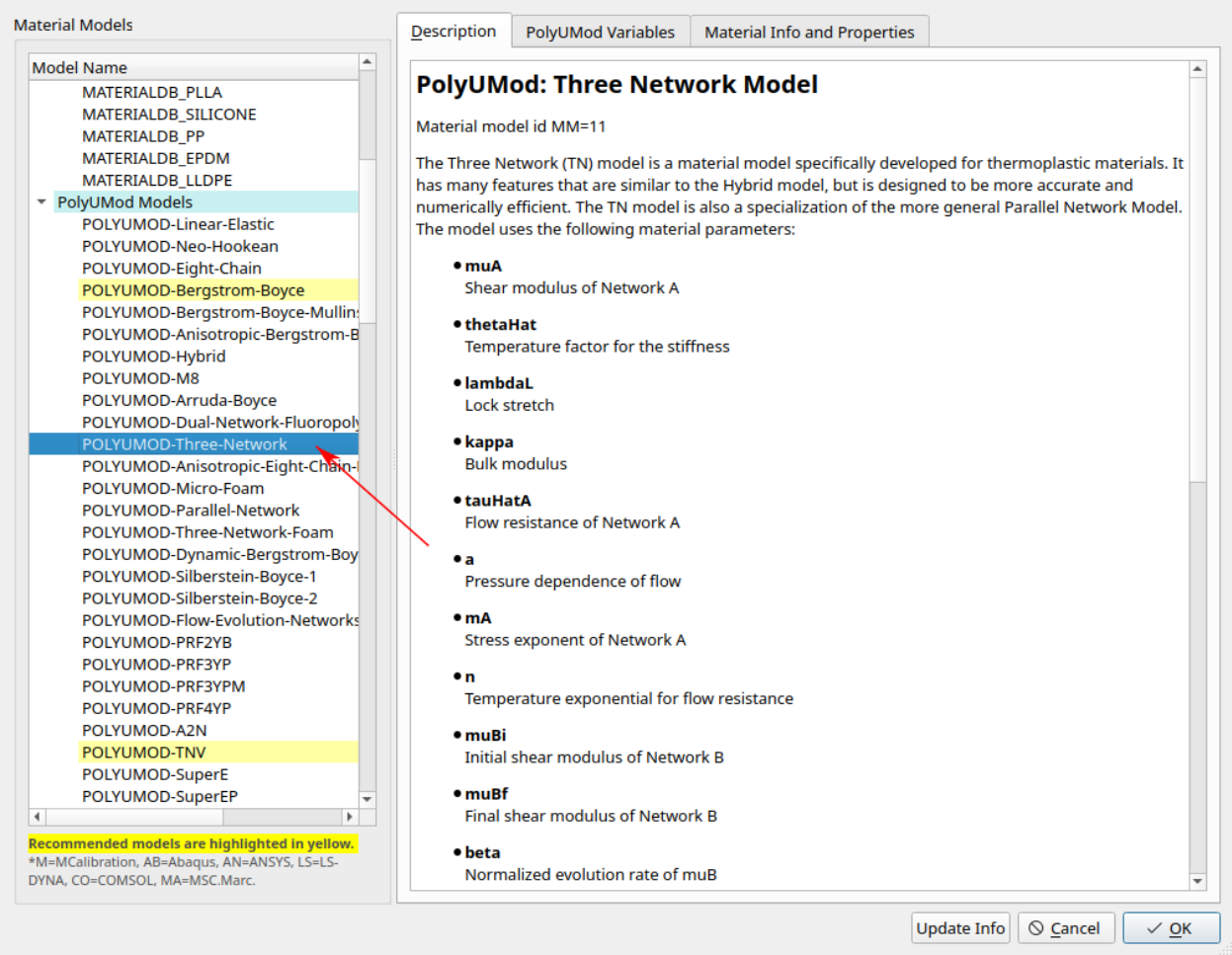
The next step is to select a suitable material model.

Click the Set Material Model... button.



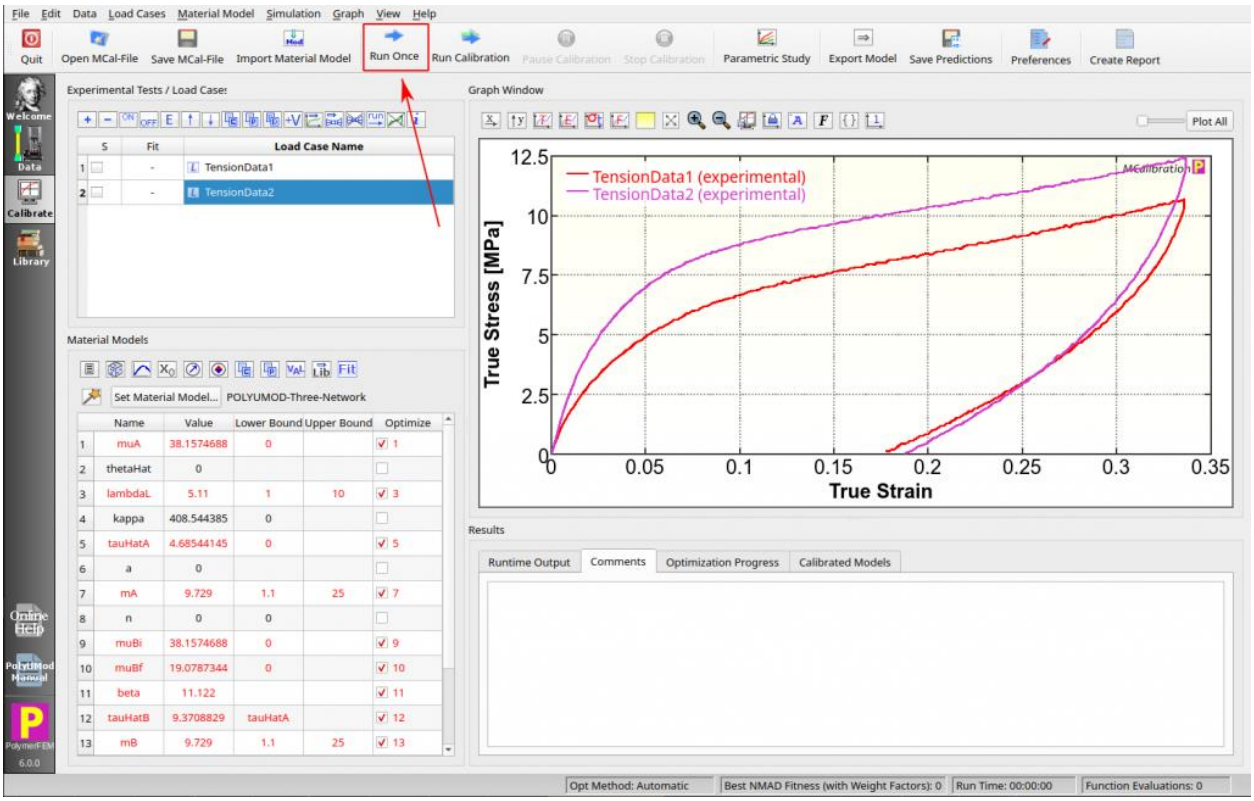
The experimental data in this example is for a medium density polyethylene (MDPE), so the Three Network (TN) model from the PolyUMod library is a good choice.

- (Optional) Fill out your information in the Material Info and Properties tab on the right.
- Select the Three-Network Model item, then click OK.



MCalibration then selects an initial guess of the material parameters based on the available experimental data.

Click Run Once to calculate the predicted stress-strain response of the current material model.

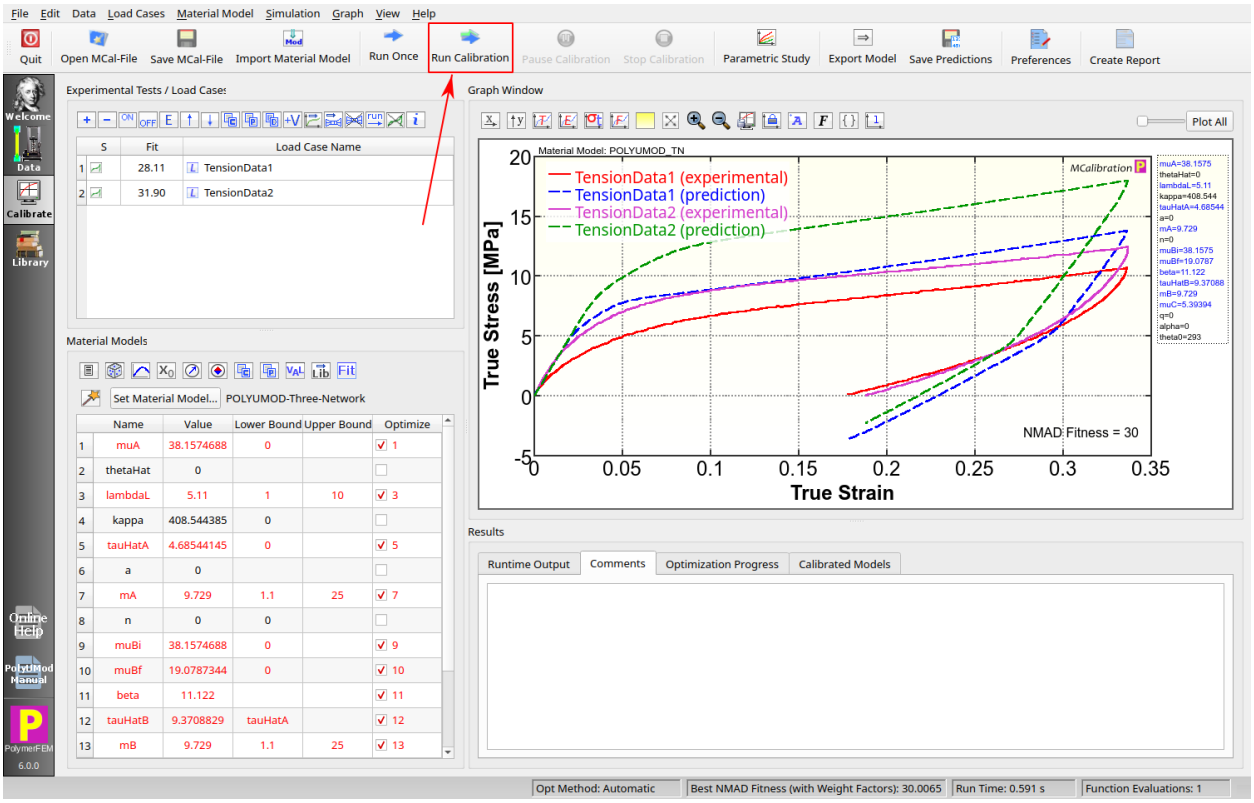

File Edit Data Load Cases Material Model Simulation Graph View Help
Quit
Open MCal-File
Save MCal-File
Import Material Model
Run Once
Run Calibration
Pause Calibration
Stop Calibration
Parametric Study
Export Model
Save Predictions
Preferences
Create Report
Welcome
Data
Calibrate
Library
Online Help
PolyMod Manual
PolymerFEM 6.0.0
Experimental Tests / Load Cases
S
Fit
Load Case Name
1 - TensionData1
2 - TensionData2
Material Models
Set Material Model...
POLYUMOD-Three-Network
Name Value Lower Bound Upper Bound Optimize
1 muA 38.1574688 0 1
2 thetaHat 0
3 lambdaL 5.11 1 10 3
4 kappa 408.544385 0
5 tauHatA 4.68544145 0 5
6 a 0
7 mA 9.729 1.1 25 7
8 n 0 0
9 muBi 38.1574688 0 9
10 muBf 19.0787344 0 10
11 beta 11.122 11
12 tauHatB 9.3708829 tauHatA 12
13 mB 9.729 1.1 25 13
Graph Window
Plot All
TensionData1 (experimental)
TensionData2 (experimental)
MCalibration
True Stress [MPa]
True Strain
12.5
10
7.5
5
2.5
0
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
Results
Runtime Output
Comments
Optimization Progress
Calibrated Models
Opt Method: Automatic
Best NMAD Fitness (with Weight Factors): 0
Run Time: 00:00:00
Function Evaluations: 0

Preliminary Calibration

The predicted stress-strain curves are shown in here blue and green, and the experimental curves are shown in red and purple. Note that the material model has not been calibrated yet. The results shown here are just the predictions from the initial guess.

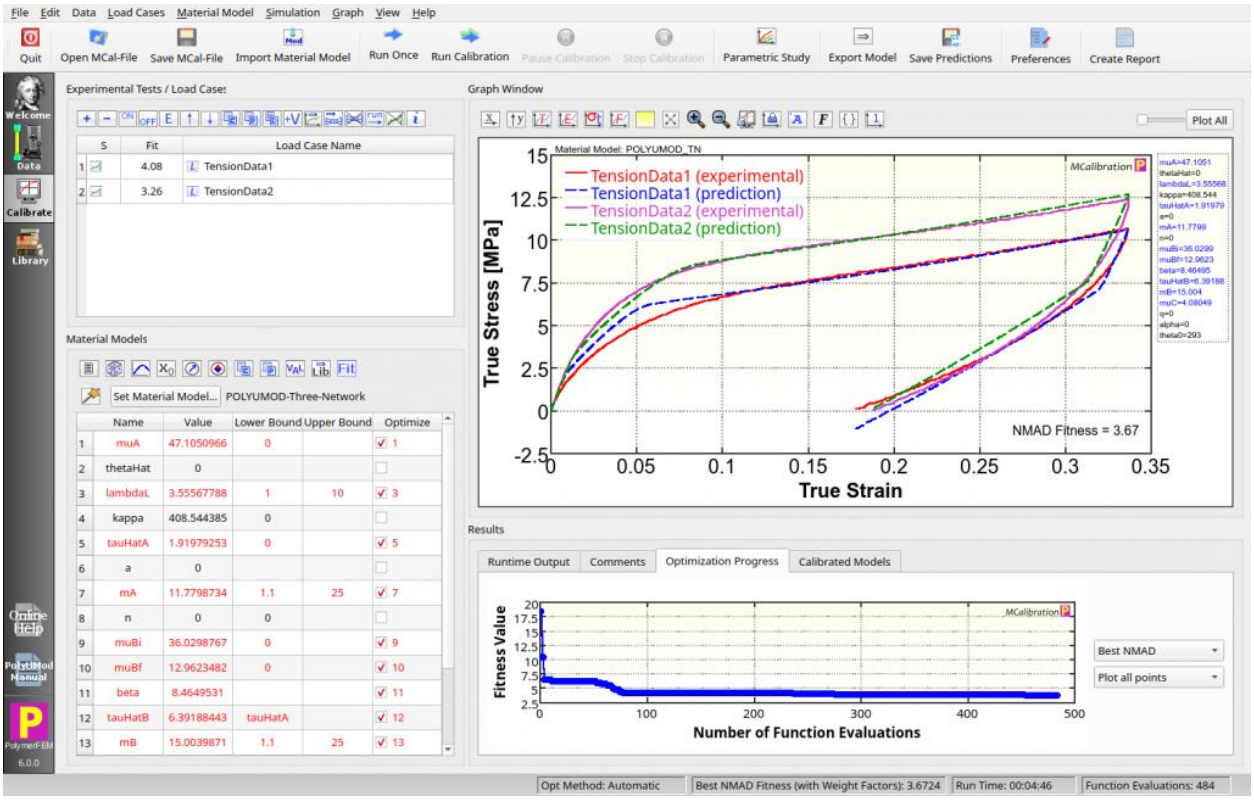
Each material parameter can either be fixed or part of the optimization. The Optimize column specifies the state of the parameters. All parameters with a non-zero positive value are included in the optimization. If two parameters are given the same optimization value then those two parameters are forced to have the same (unknown) value.

- Click Save File to save the current calibration file.
- Click Run Calibration to start optimizing the material parameters.



Model Calibration

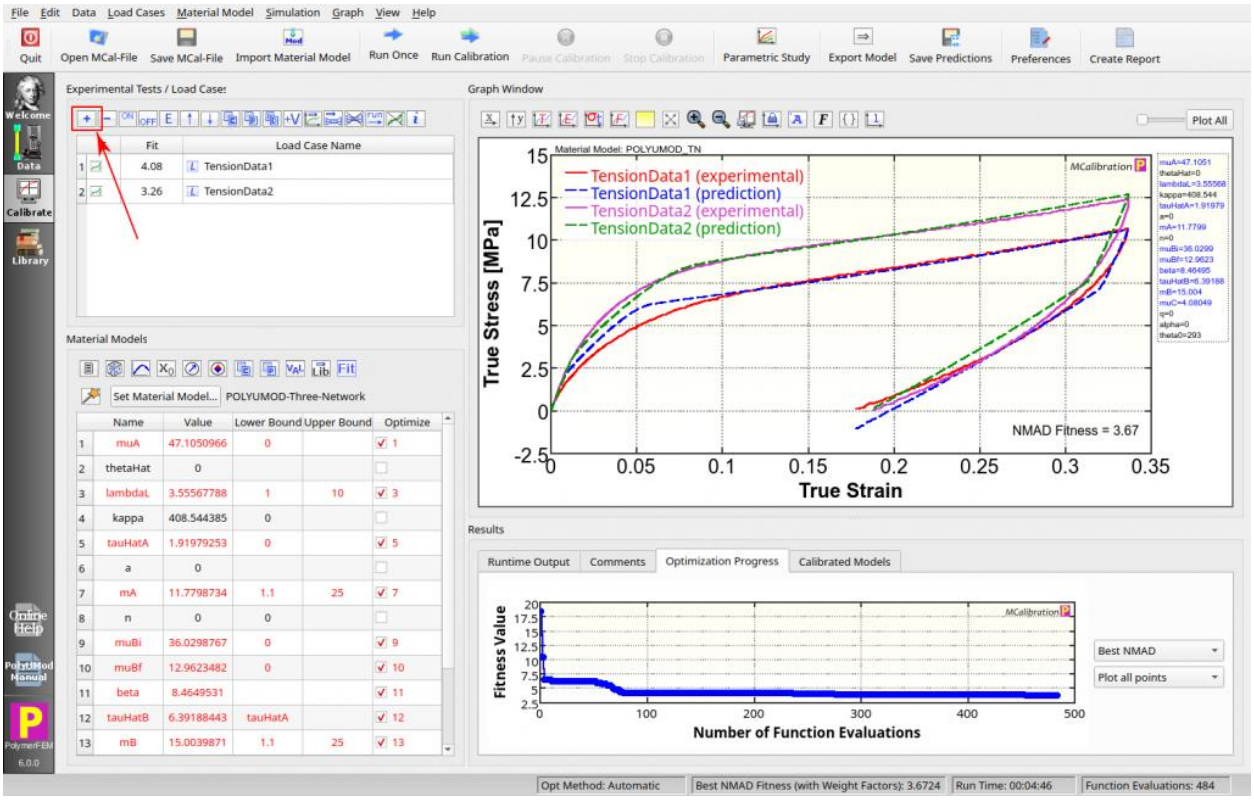
We manually stopped the calibration after a few minutes. At this point the error in the material model predictions (NMAD Fitness) is less than 3%.



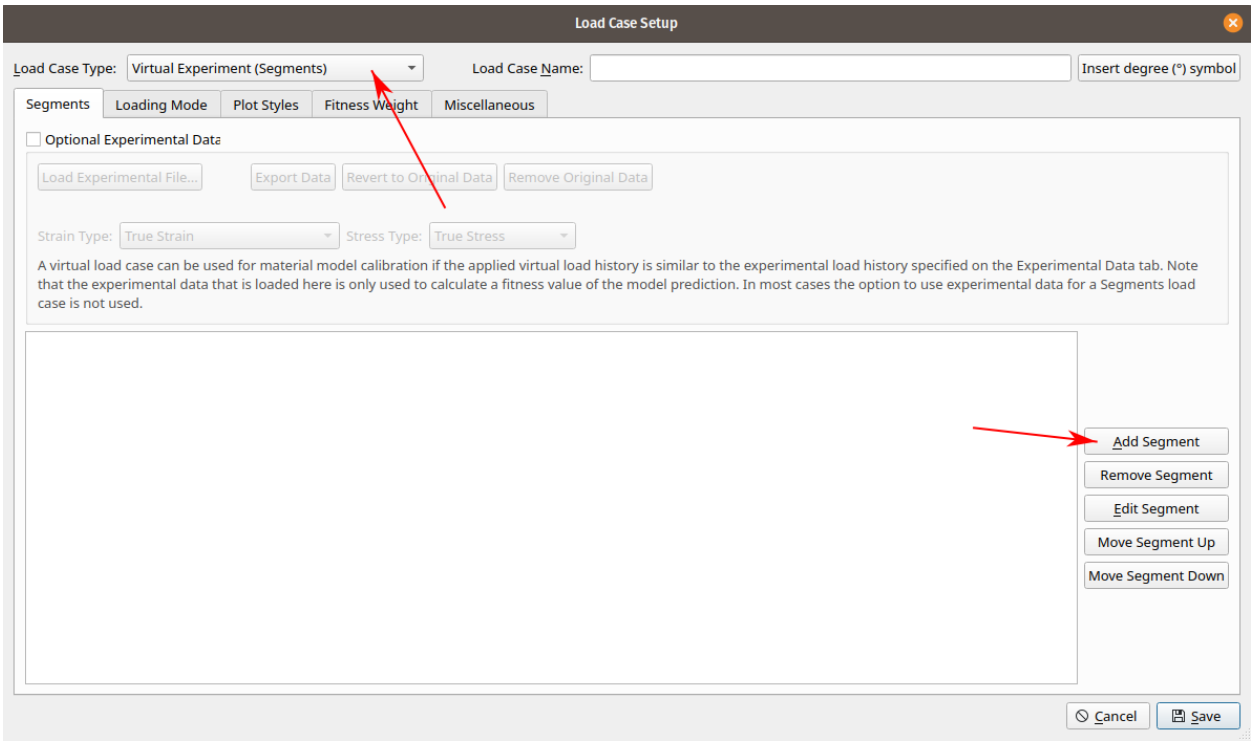
Virtual Stress Relaxation

Sometimes it is useful to examine how a material model behaves under conditions that have not been experimentally tested. Here we will perform a virtual uniaxial compression experiment to an engineering strain of -0.1 followed by 60 seconds of relaxation.

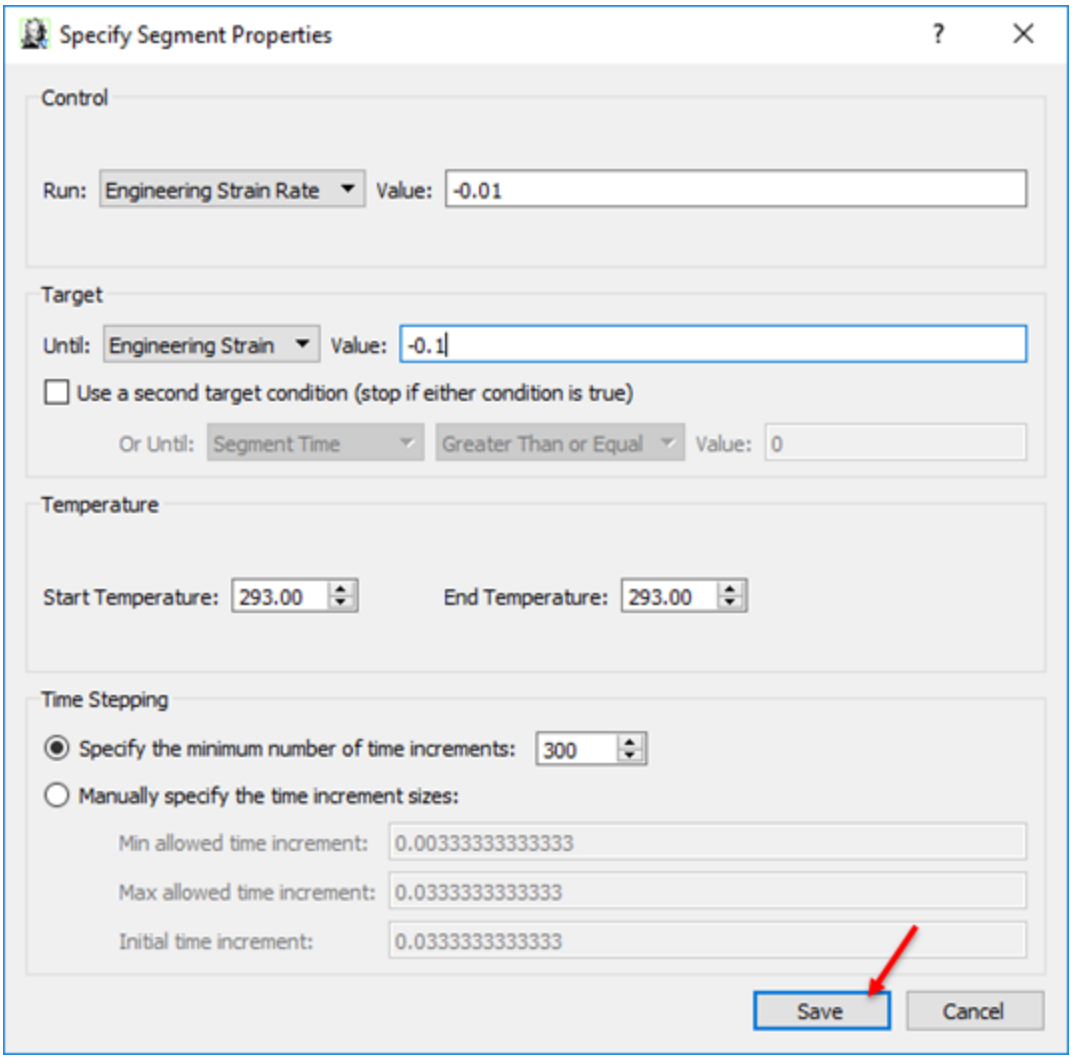
- Click on the + button to setup the virtual experiment.



- Select Virtual Experiment (Segments) from the load case type drop down list.
- Click Add Segment.

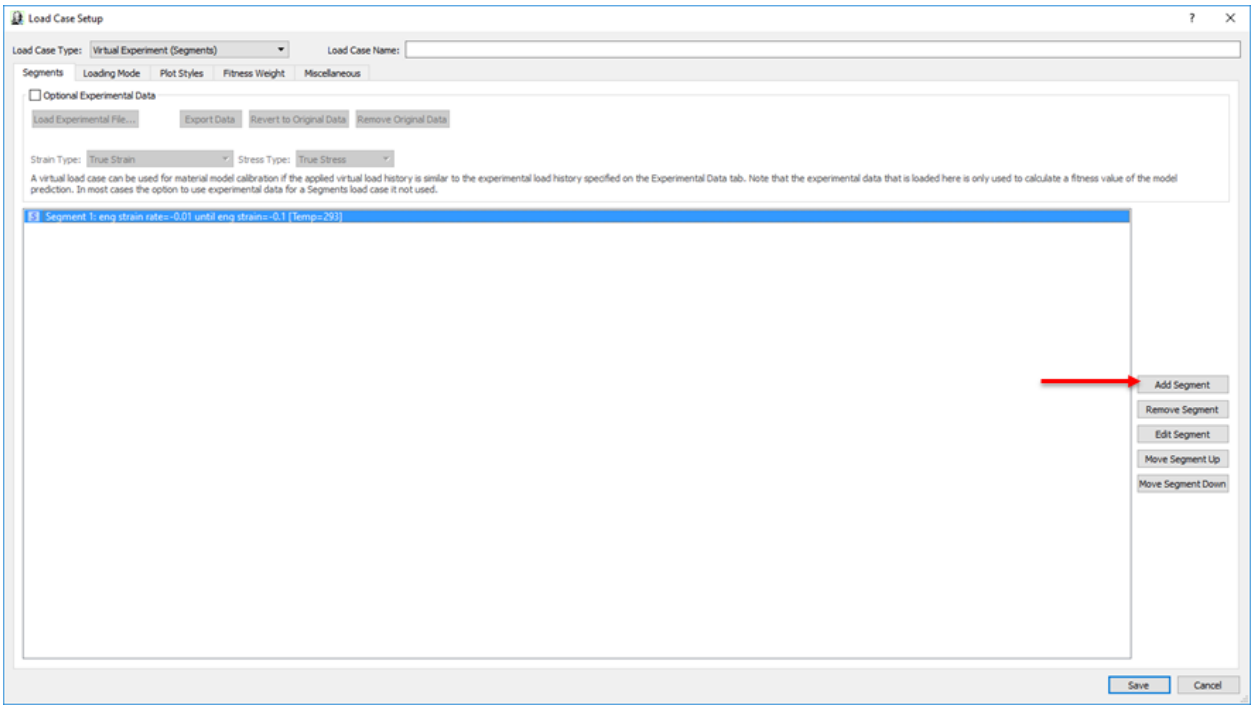


- Specify the target strain rate and strain value. This specifies the stress relaxation pre-load.
- Then click the Save button.



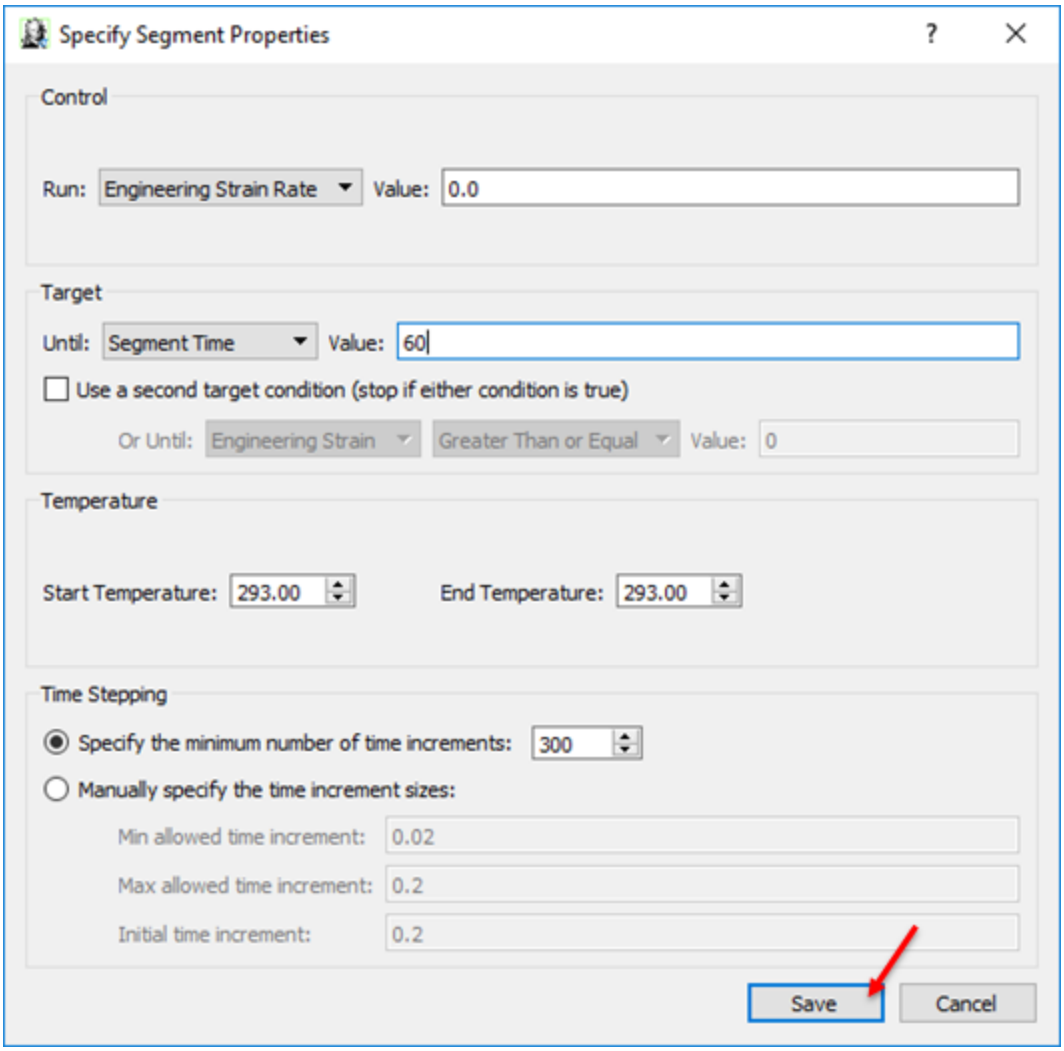
The load case dialog now contains the first loading segment of our virtual experiment. Next, we need to create the second stress relaxation segment.

- Click the Add Segment button.



The second loading segment has constant strain for 60 seconds.

- Click Save.

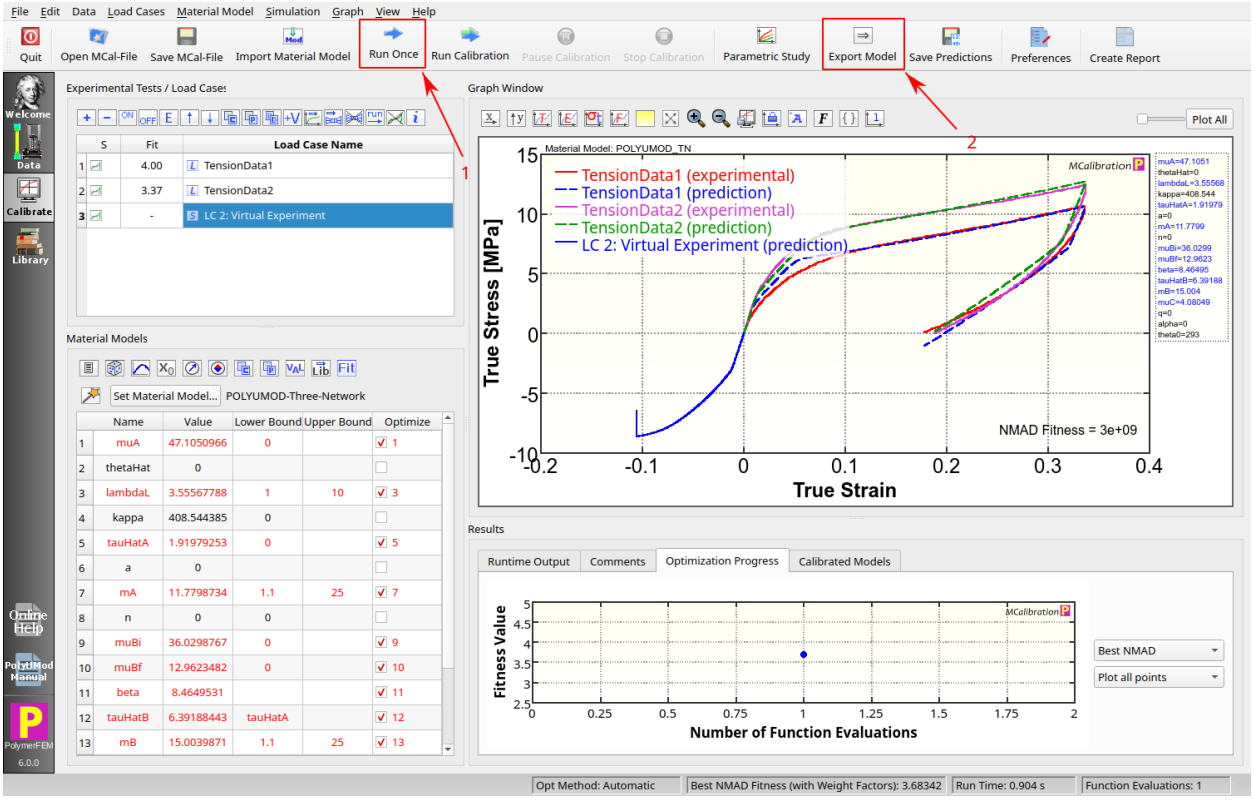


Back in the main window click the Run Once button to evaluate the new load virtual experiment.

We see that the predicted stress relaxes about 30% in 60 seconds.

The next step is to export the calibrated model to a FE program.

- Click the Export Model button.


File Edit Data Load Cases Material Model Simulation Graph View Help
Quit
Open MCal-File
Save MCal-File
Import Material Model
Run Once
Run Calibration
Pause Calibration
Stop Calibration
Parametric Study
Export Model
Save Predictions
Preferences
Create Report
Welcome
Data
Calibrate
Library
Online Help
PolyMod Manual
PolymerFEM 6.0.0
Experimental Tests / Load Cases
S
Fit
Load Case Name
1 4.00 TensionData1
2 3.37 TensionData2
3 - LC 2: Virtual Experiment
1
Material Models
Set Material Model...
POLYUMOD-Three-Network
Name Value Lower Bound Upper Bound Optimize
1 muA 47.1050966 0 1
2 thetaHat 0 0
3 lambdaL 3.55567788 1 10 3
4 kappa 408.544385 0 0
5 tauHatA 1.91979253 0 5
6 a 0 0
7 mA 11.7798734 1.1 25 7
8 n 0 0 0
9 muBi 36.0298767 0 9
10 muBf 12.9623482 0 10
11 beta 8.4649531 11
12 tauHatB 6.39188443 tauHatA 12
13 mB 15.0039871 1.1 25 13
Graph Window
2
Plot All
Material Model: POLYUMOD_TN
MCalibration
TensionData1 (experimental)
TensionData1 (prediction)
TensionData2 (experimental)
TensionData2 (prediction)
LC 2: Virtual Experiment (prediction)
muA=47.1051
thetaHat=0
lambdaL=3.55568
kappa=408.544
tauHatA=1.91979
a=0
mA=11.7799
n=0
muBi=36.0299
muBf=12.9623
beta=8.46495
tauHatB=6.39188
mB=15.004
muC=4.08049
q=0
alpha=0
theta0=293
NMAD Fitness = 3e+09
True Stress [MPa]
True Strain
Results
Runtime Output
Comments
Optimization Progress
Calibrated Models
Fitness Value
Number of Function Evaluations
Best NMAD
Plot all points
Opt Method: Automatic
Best NMAD Fitness (with Weight Factors): 3.68342
Run Time: 0.904 s
Function Evaluations: 1

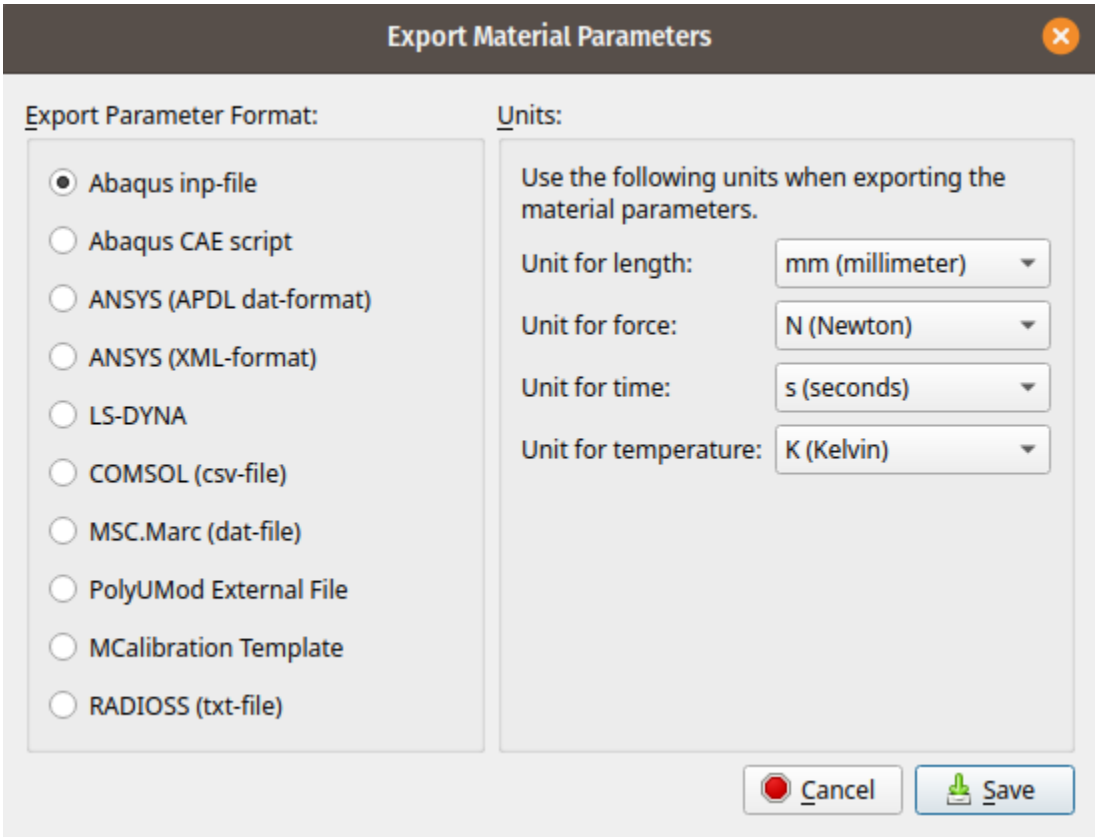
Export Material Model

To export the material model to Abaqus/CAE select Abaqus CAE script or Abaqus inp-file, and click Save.

To export the material model to ANSYS select ANSYS (APDL or XML format), and click Save.

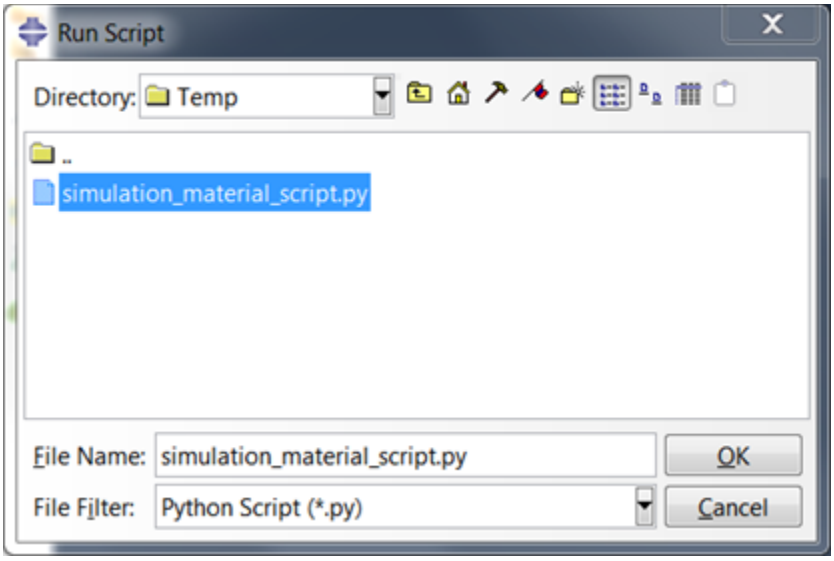
The material model can also be exported to MSC.Marc, LS-DYNA, Radioss, and COMSOL formats.

You may modify the saved units system from this dialog.



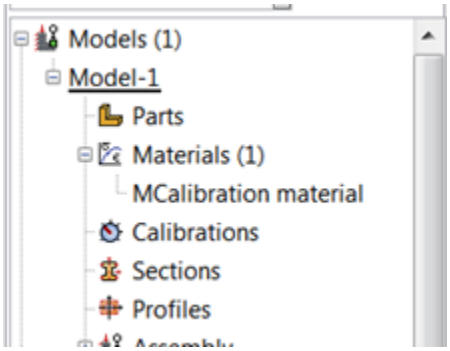
Import Model Into Abaqus/CAE

In CAE select Run Script from the File Menu, then select the script file that was created by MCalibration.



The model tree then contains the calibrated material model.

For more examples, see also Validating the Installation (PolyUMod for Abaqus)

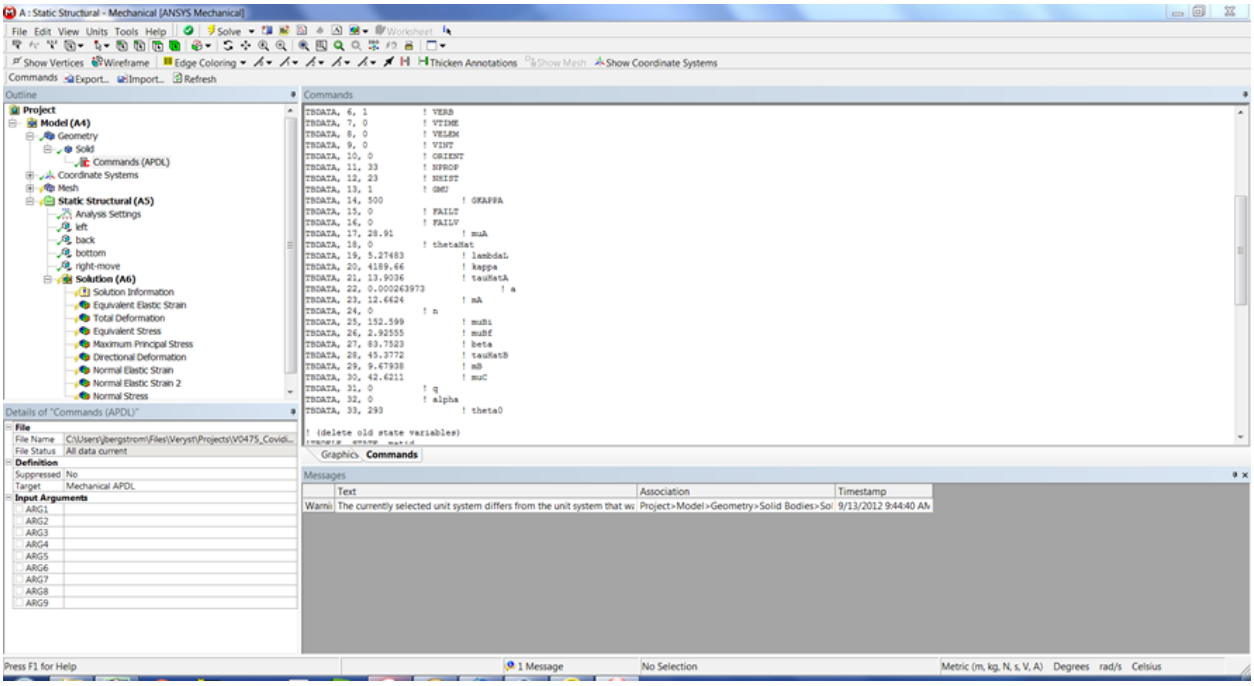


Import the Model Into ANSYS WB

Read the .dat file exported from MCalibration into ANSYS Mechanical as a command under Geometry → Solid or read the .xml file into Engineering Data.

Make sure non-linear geometries are enabled in Mechanical.

See also Using PolyUMod with ANSYS Workbench.



MCalibration File Format

MCalibration saves the simulation information in a file with the extension mcal.

- The mcal-file is a XML file that can be edited using a text editor.
- The file contains all experimental data and information about the different load cases and material models.
- Since the mcal-file contains the experimental data, the file can be moved to a new directory and still work.
- The experimental data that is stored in the mcal-file includes both the original data and the current version of the data (if the data has been modified in the Data tab).
- The original and modified experimental data sets can be exported to separate data files if needed.

MCalibration Summary

- MCalibration is an easy-to-use tool that can calibrate many different material models.
- One of the most powerful features of MCalibration is that it can use almost any combination of experimental data, e.g. tension, compression, shear, biaxial, triaxial, stress relaxation, creep, DMA, Poisson's ratio, etc.
- MCalibration can also use direct finite element simulations of more complicated experimental tests to calibrate a material model.