Ecole Polytechnique Fédérale de Lausanne

Module 2

Material Model for silicone based on uni-axial traction test

Joao Barini Ramos
Nathan Soury-Lavergne
Syrielle Wicki
James Ziadeh

ME-412

Introduction

The goal is to characterize the silicone material (Elite Double 32) that is used for the experiments. Indeed for any model chosen (elastic model or a hyperelastic one for example), it is crucial to experimentally determine material constants or to describe how stress is related to strain somehow. The model could be later used in a FEM software in order to model any configuration using Elite Double 32 subjected to traction loads.

Method

The approach is based on a tutorial by ANSYS [1].

A 45mm x 90mm x (1.26mm) [1] silicone (Elite Double 32) has been created using a mold and by respecting the supplier guidelines concerning proportion of base and catalyst. This plate is then subjected to traction until rupture using a *INSTRON 6848 Micro-tester* traction machine.

An effective length of 42.36mm is used (corresponds to the distance between the two supports of the traction machine).

Strain is calculated by dividing the elongation by the original length. We also highlight that the an engineering stress is computed. Therefore the load is divided by the original cross section of the sample and not by its real value.

It is than possible to use ANSYS to find a suitable model characterizing the material. When a model is chosen, ANSYS finds the best curve that fits the dataset provided (stress and strain points) by minimizing the squared error according to Eq.1.

$$Error = \sum_{i=1}^{N} (\sigma_E^i - \sigma_M^i)^2 \quad (1)$$

where σ_E^i is the experimental stress provided and σ_M^i the one proposed.

The final squared error is the *Residual* of the model from the least-square function. The lowest this parameter, the better the model is.

The model is than tested by imposing a single element model configuration on the FEM software. The idea is to constrain x, y and z displacements of the faces composing the base of the plate and displace the upper section on the y axis. Thus the area being displaced remains the same. Indeed, a force reaction is computed using the solution. The area of the cross section of the sample stays constant due to the imposed boundary conditions. Therefore, the numerical value provided by the FEM solution also corresponds to an engineering stress. The displacement in the y direction also corresponds to an engineering strain. It is therefore possible to compare them to the experimental stress and strain also computed using an initial cross section.

This is achieved by setting the mesh size to be equal to 45mm, by enabling large deformations on ANSYS, switching off Auto time stepping and by setting the number of substeps to 10 (10 points are therefore calculated).

Results

The results of the traction test and the a curve fitting are presented on Fig. 3. The proposed fitting uses the Mooney Rivlin 3 model [2]. This model yields to a residual of 0.1525 Pa^2 which seems reasonable. Visual inspection also indicates a good approximation.

Material Constant C01: 149223 Pa
Material Constant C10: 14261 Pa
Material Constant C11: 19980 Pa

We can easily see that a linear elasticity model is not suitable for the whole span of strain values proposed. If we assume a linear elastic behaviour for up to 10% deformation, a Young's Modulus of E=817858 Pa can be computed using a nearly perfect incompressible isotropic material assumption ($\nu = 0.49$). The stress vs strain curve using linear elasticity up to 10% deformation are plotted on Fig. 1 A Residual of 520618021 Pa^2 is found.

[1] The 1.26mm thickness is calculated by averaging 2 measurements

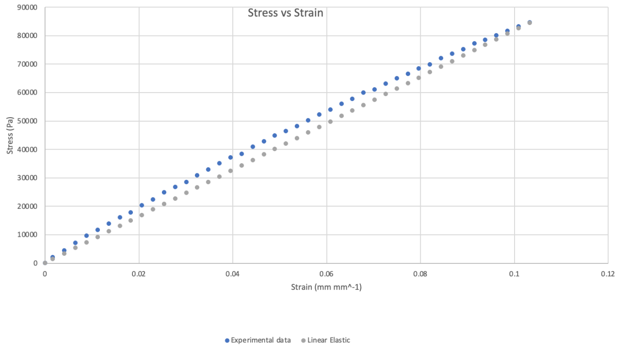

Figure 1: Engineering stress vs engineering strain curves using isotropic linear elasticity model with E=817858 Pa and $\nu = 0.49$ (in grey) and experimental data (in blue).

The stress vs strain diagram for the single element model computed using FEM is presented on Fig. 2. The experimental data is also presented. It seems a good fit.

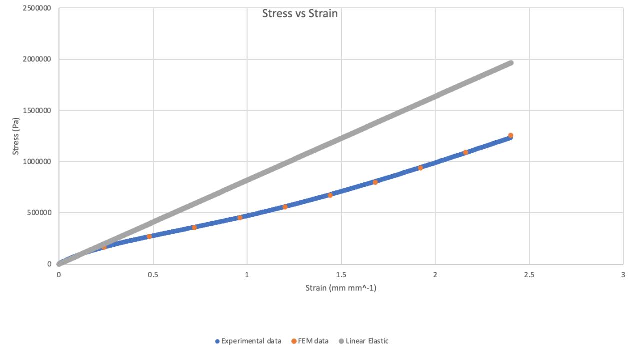

Figure 2: Engineering stress vs engineering strain curves for single element model using FEM, isotropic linear elastic and experimental data

Improvements

There is room for improvement by performing more traction tests and averaging the results as well as taking more thickness measurements since those are crucial when determining stress. It is important to highlight that in order to have a more accurate model, it would have been necessary to perform experiments also for biaxial, shear and volumetric loads. Therefore the computed results are only valid for tensile loads.

References

[1] ANSYS. *Performing Curve Fitting for Hyperelastic Models in Ansys Mechanical*. 2020. URL: https://www.youtube.com/watch?v=K0rIwPEi0mg.

[2] WELSIM - Engineering Simulation. *Mooney-Rivlin hyperelastic model for nonlinear finite element analysis*. 2020. URL: https://getwelsim.medium.com/mooney-rivlin-hyperelastic-model-for-nonlinear-finite-element-analysis-b0a9a0459e98.

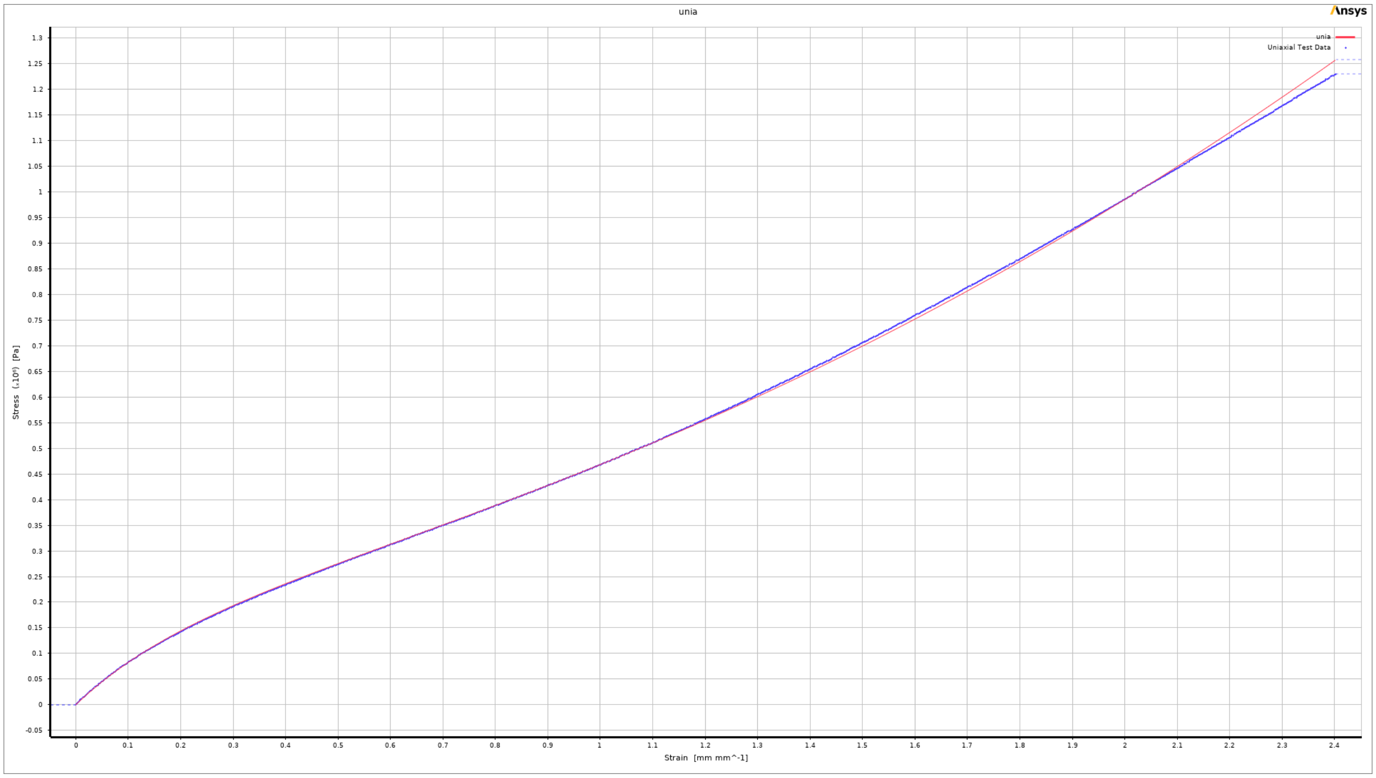

Figure 3: In red: Stress vs Strain curve obtained using Mooney Rivlin 3 parameter curve fitting in ANSYS. In blue: experimental stress vs strain curve obtained through traction test. Both stresses and strains are compueted using the engineering approach (Original cross section). The data is computed from a strain of 0 up to 2.5