Developing a Constitutive Model for a Highly Deformable Latex Rubber with Implementations in Lagrangian Frameworks

Christopher A. Basciano[1], Marcus Rademacher[1], Patrick Downie[1], Anita Bestelmeyer[1], Brian Croop[2], Jorgen Bergstrom[3]

[1]BD Corporate CAE, 21 Davis Drive, Research Triangle Park, NC 27709
[2]DatapointLabs, 95 Brown Rd. #102, Ithaca, NY 14850
[3]Veryst Engineering LLC, 47 Kearney Rd, Needham Heights, MA 02494

Abstract: To improve product design and enhance the realism of BD's advanced finite element models, the development of a constitutive model capable of predicting mechanical behavior at high deformations was initiated for a commercially available latex rubber. A set of baseline experiments was conducted to determine the isothermal mechanical response(s) of the latex rubber. The experiments were designed to provide fundamental insight into any viscoelastic, hyperelastic, and Mullins effects exhibited by the material. A constitutive model was constructed for specific load cases from the aforementioned experimental data. Attention was given to the model's primary use case, numerical stability, and its ability to perform at high deformation ranges. A second set of experiments was then conducted, aimed at characterizing the failure of the rubber and providing a set of experimental data to validate the material's constitutive model. The first round of experiments revealed that the material's stress vs. strain curves held limited dependence on the applied loading rate, while the failure stress and strain had a moderate dependence. Additionally, cyclic loading and unloading of the material was shown to have a prominent effect on the failure stress and strain. Based on the model's intended use case and baseline mechanical response, a hyperelastic constitutive model was calibrated to the monotonic, uniaxial tension data. The model was then implemented in Lagrangian and Eulerian geometries and compared against the second set of experimental data. The constitutive model exhibits good agreement to the calibration data when strain is less than 5.0, but begins to significantly underpredict the stress values at strains above 6.0. Validation studies revealed good agreement between the experimental tear test data and different implementations of the constitutive model with different boundary conditions. Work is ongoing to extend the implementations of the constitutive model to an Eulerian framework and refine the constitutive model calibration.

Keywords: Constitutive Models, Hyperelasticity, Hyperelastic Material, Tear Test, Rubber

1. Introduction

Highly-deformable rubbers are often used in medical devices and normally require nonlinear constitutive models to accurately represent the material's mechanical response. Abaqus includes a wide range of such hyperelastic material models available for implementation in finite element simulations, but care must be taken to ensure the constitutive material model provides a numerically stable formulation and accurately represents the material's response for the desired context of use. Challenges often arise in determining the appropriate constitutive model and its corresponding parameters that enable a satisfactory fit to the experimental data used for model calibration. Comparing the model against experimental datasets not used to calibrate the constitutive model is an imperative step to define the model's context of use and understand the model's predictive capabilities. In this paper, we present a study to fundamentally characterize the material's mechanical response, calibrate and select a constitutive material model, characterize the tear strength of the material, and implement the constitutive model in finite element models of the tear strength experiments.

This paper is organized into the following sections. In Section 2, we discuss the methods and methodologies for each workstream: the selection of appropriate experiments and the procedures followed for the material characterization (cf 2.1), the methodology followed for the constitutive material model selection and calibration (cf 2.2), and the constitutive model implementation in Abaqus/Standard and Abaqus/Explicit. Section 3 illustrates and discusses the results from each workstream: the experimental tests (cf 3.1), the constitutive material model calibration and selection (cf 3.2), and the constitutive model implementation in Abaqus/Standard and Abaqus/Explicit (cf 3.3). The paper concludes with comments of continuing work (Section 4) and overall conclusions of the aforementioned studies (Section 5).

2. Methods

2.1 Experimental material characterization

The selected material for this work was commercially available latex rubber. The material came as a single sheet 36 inches long, 6 inches wide, and 0.04 inches thick. Test specimens were punched or cut into the desired forms as needed. A suite of tests were conducted to broadly explore and characterize different aspects of the material response. The tests included i) monotonic uniaxial tensile tests at different strain rates, ii) cyclic uniaxial tensile tests at different strain rates, iii) Mullins effect uniaxial tensile tests, iv) cyclic compression tests, and v) creep relaxation DMA testing. No pre-cycling was administered to the material to represent the end use of the material. An Instron 5566 universal material testing system was used for all of the tension and compression tests. Tensile experiments were conducted using standard ASTM D412 Type C tensile bars and video extensometry was used for the strain measurements. Compression experiments were conducted after ASTM D-695 2010 and used multiple 1 inch diameter disks that were stacked on top of one another between lubricated plates 0.331 inches apart. DMA testing was performed at two different strain rates and the strain of 100% was held constant for 8 hours.

Tearing tests were also conducted to supplement the previously discussed experimental dataset with material failure data. The ASTM D624 Bowtie/Type C geometry was used for the tearing tests and experiments were conducted at two different strain rates. An important note is that the tearing data was not used to determine the constitutive model parameters and could be used as a validation dataset for the constitutive model parameter calibration.

2.2 Material model calibration

After reviewing the material test data and the material model's intended use case (i.e., loading conditions), the monotonic, tensile loading was deemed the most representative for the model's context of use and the compression, DMA, and cyclic tensile testing data was not used to calibrate the material model. Specific focus was given to numerical stability and achieving a good fit at multiple regions along the stress vs strain curve. Multiple material models were considered including the Bergstrom-Boyce, Yeoh, Ogden, and reduced polynomial strain energy functions. Details on the mathematical formulations of hyperelasticity and strain energy functions each model can be found in (Holzapfel, 2000), the PolyUMod User's Manual, and the Abaqus analysis guide. The calibration tools included Abaqus/CAE and MCalibration (Veryst Engineering, LLC, Needham Heights, MA). The net error between the experimental data and constitutive model predictions was calculated using the normalized mean absolute difference (NMAD) between the experimental data. An iterative error minimization scheme semi-automatically determined the model parameters that would minimize the net error between the experimental data and model predictions.

2.3 Material model implementations

The material model was implemented in static Abaqus/Standard v14.1 and quasi-static Abaqus/Explicit v14.1 simulations of the ASTM D624 Type C experimental tearing test, which were not used to calibrate the material model. All simulations were displacement-driven and the Abaqus/Explicit simulations had a load step time, displacement amplitude, and target stable time increment to ensure minimal mass scaling and kinetic effects. The element types used in the simulations were C3D8H elements for Abaqus/Standard simulations and first-order C3D8R elements with enhanced hourglass control for Abaqus/Explicit simulations. A mesh refinement study was conducted by comparing the calculated force vs. displacement curves or different element densities across the Type C geometry.

A variety of boundary conditions were tested to assess the influence of different gripping conditions and the enforcement of symmetry. The gripping regions were kinematically coupled to reference points outside the geometry to enable efficient load specification and force vs. displacement post-processing. Figure 1 illustrates the different Lagrangian formulations of the Type C tear test simulation.

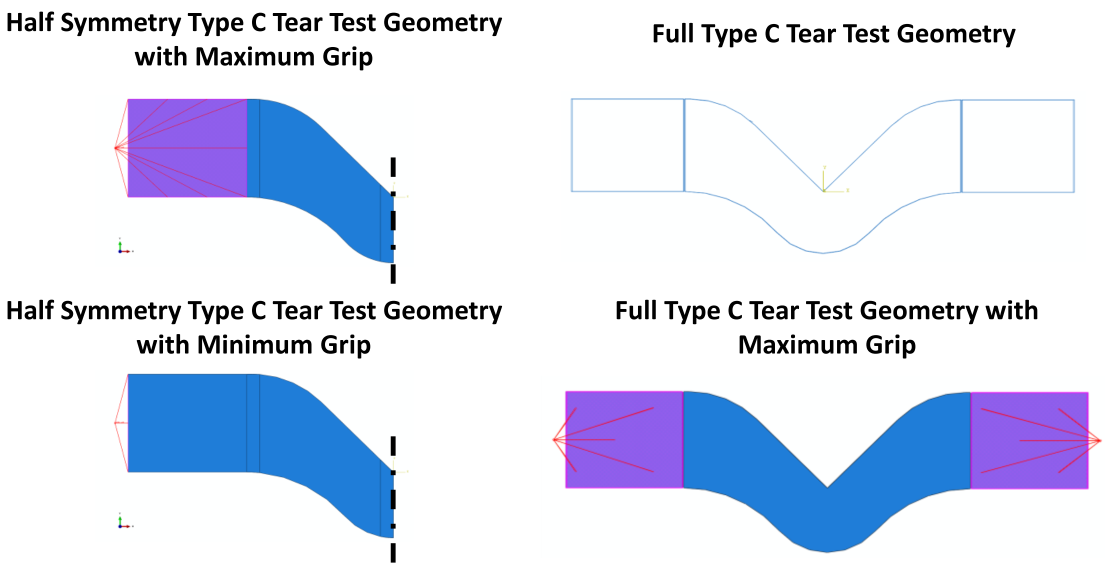

Figure 1. Kinematic coupling and symmetry boundary conditions of the Type C tear simulation for maximum and minimum gripping scenarios

3. Results and Discussion

3.1 Experimental material response

Figure 3 illustrates representative tensile data collected for the latex rubber for three different strain rates on the orders of 0.001, 0.01, and 0.1 /s. The material's mechanical response exhibited a relatively low sensitivity to strain rate and a limited energy loss during loading and unloading cycles. Ultimate strains at failure were around 1000% for monotonic uniaxial tension. Whereas, the Mullins test data revealed a noticeable sensitivity to cyclic load history with a failure strain around 750% and a reduction in the ultimate stress when compared to the monotonic uniaxial tension data.

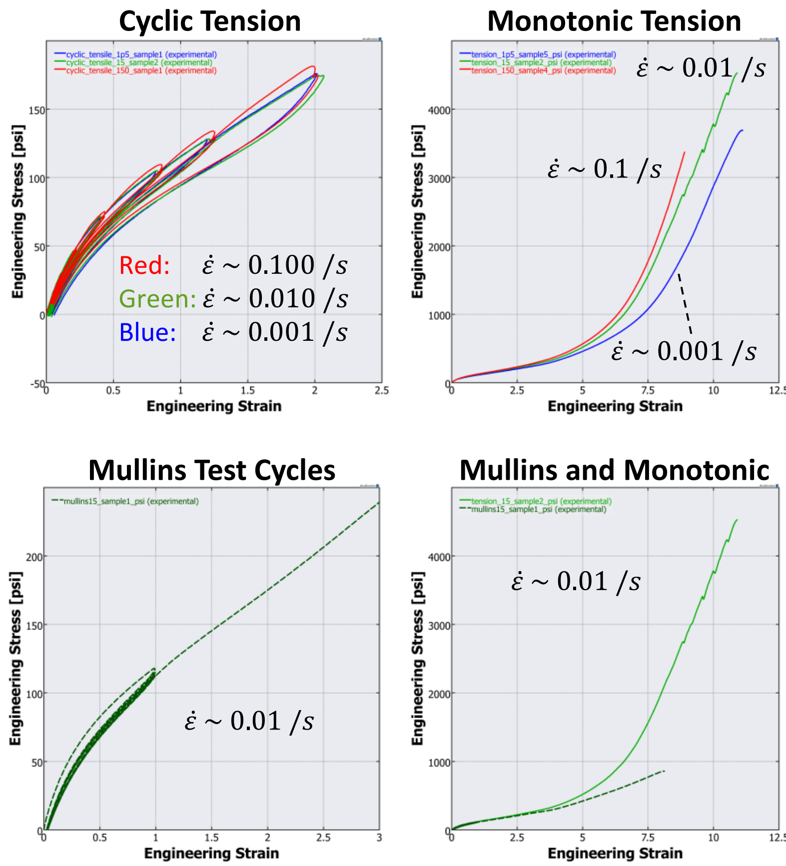

Figure 3. Representative experimental tensile data for the latex rubber.

Figure 4 illustrates representative tearing test data for the ASTM D624 Type C geometry for different extension rates an order of magnitude apart (0.315 and 0.0315 in/s.). An important note is that the extension rate had a noticeable dependence on the displacement and force at material tearing/failure.

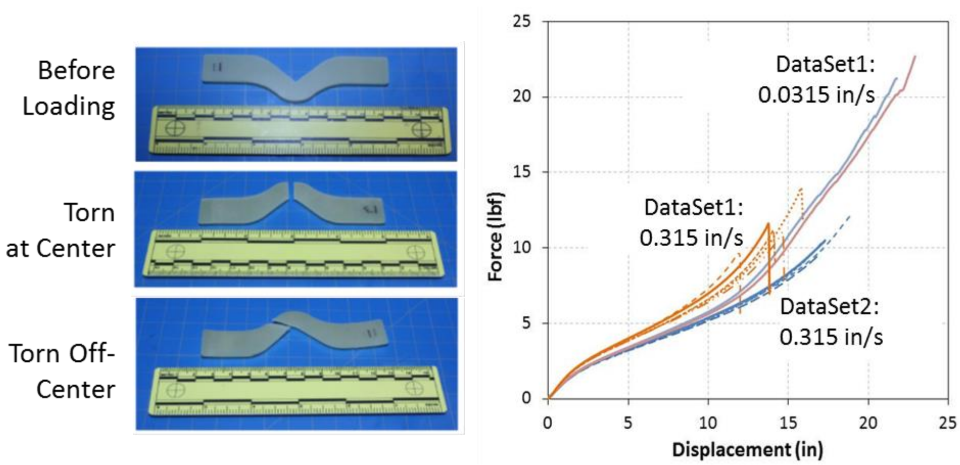

Figure 4. Experimental tearing test data for the latex rubber.

3.2 Material model calibration

As previously mentioned, multiple models were considered for material calibration to the monotonic uniaxial tension experimental data. Due to the material's limited viscoelastic effects and high levels of deformation, hyperelastic models were deemed suitable to represent the primary characteristics of material's mechanical response. The Yeoh model was selected due to its simplicity, overall numerical stability, and enhanced convergence properties at high levels of element distortion. Figure 5 illustrates the fit between the Yeoh model and the experimental data.

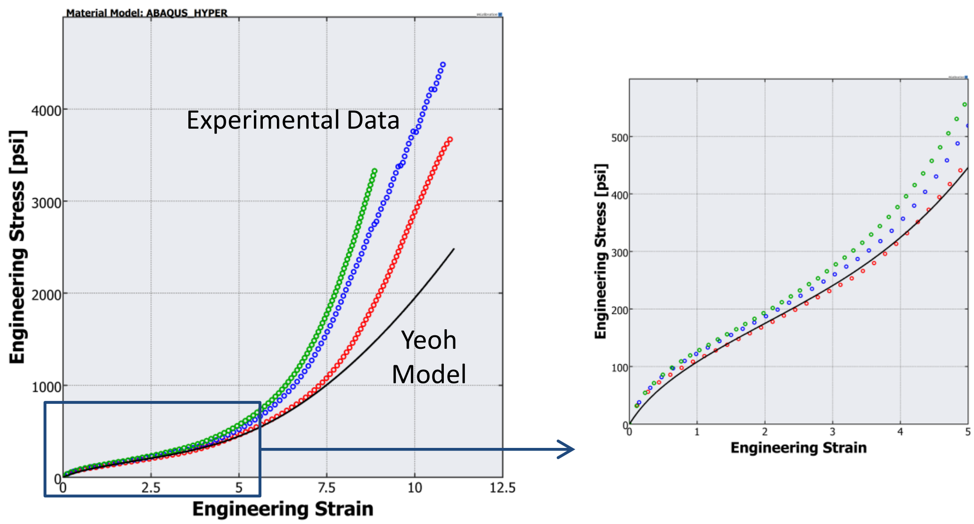

Figure 5. Calibration of the Yeoh model parameters to the monotonic uniaxial tension data

The model parameters were selected to ensure the existence of real roots for all possible shear modulus values according to the Yeoh model parameter restrictions outlined in (Holzpafel, 2000). An important note of the model's deficiency is the material model does not exhibit the same level of stress stiffening as the experimental data at engineering strains above 5.0 (500%). Decreasing the discrepancy at higher strains is an ongoing effort for material calibration refinement, but the model matches the data well for engineering strains below 5.0.

3.3 Material model implementations in Abaqus

The mesh refinement analysis revealed force vs displacement curves that showed no changes for the different element densities. The medium-grade mesh density was thus selected as the base mesh density for all simulations. Whereas, a comparison of the different grip geometries yielded significantly different force vs. displacement curves and also local deformation gradients for both Abaqus/Standard and Abaqus/Explicit simulations. Figure 6 illustrates the von Mises stress contours for the different grip-types in Abaqus/Explicit.

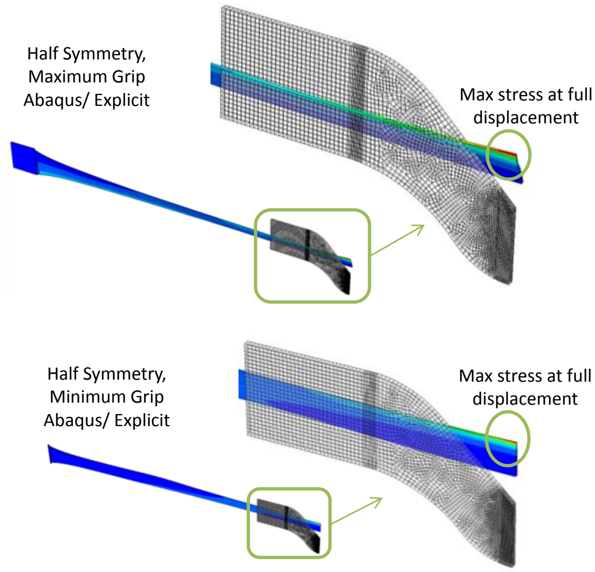

Figure 6. Stress contours for the maximum and minimum grips in Abaqus/Explicit.

An important note is the visibly different deformation gradients at the maximum stress region for the two different simulations, where the only difference between the simulations is the grip boundary conditions. For corresponding grip types, no visible difference was observed between the calculated force vs displacement curves from Abaqus/Standard and Abaqus/Explicit. Moreover, when compared with the experimental data, the two different grip conditions straddle two sets of experimental data that were collected at two different testing laboratories. Figure 7 depicts the different Abaqus simulations' force vs displacement to the experimental data. Note that Figure 7 contains data from Abaqus/Standard and Abaqus/Explicit simulations but their resulting curves lie on top of one another.

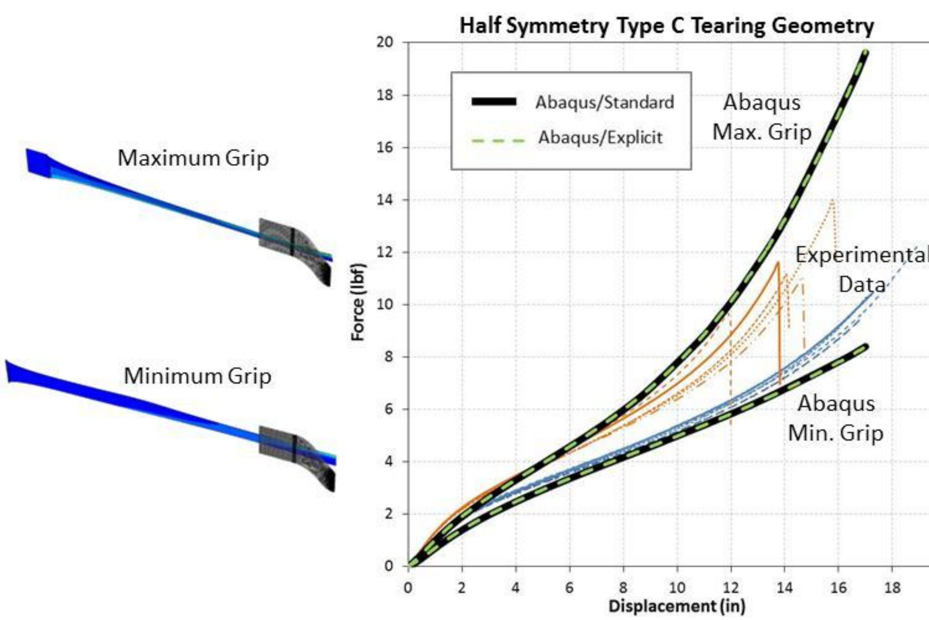

Figure 7. Comparison between the force vs displacement curves from Abaqus/Standard and Abaqus/Explicit simulations with different grip conditions and two different experimental datasets.

The results in Figures 6 and 7 are for half symmetry geometries. To ensure the constitutive model would maintain stability in full geometry simulations, a full geometry was created and run with the maximum grip condition in Abaqus/Explicit. Figure 8 illustrates the close match between the force vs displacement curves for the full and half symmetry geometry.

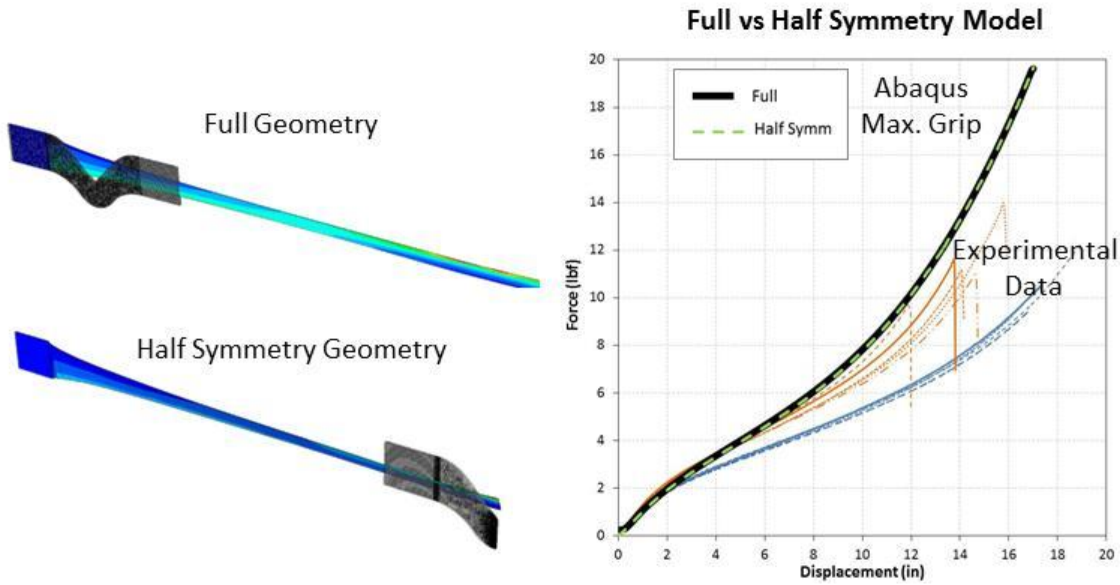

Figure 8. Comparison between the force vs displacement curves from half-symmetry and full-geometries with maximum grip conditions in Abaqus/Explicit and two different experimental datasets.

4. Ongoing Work

Significant work has been completed on developing a constitutive model for highly deformable latex rubber, but opportunities exist to further enhance the constitutive model and its implementations. The first is to refine the model's mechanical response at higher strains. To accomplish this task, we are trying different calibration procedures such as fitting the model parameters in a piecewise fashion. Another approach is to consider models with higher order functions of the first invariant. Additional model enhancements include developing material failure model(s) that have a dependence on loading rate as shown in the Type C tearing test experimental data. The last piece of ongoing work is to implement the constitutive model in a coupled Eulerian Lagrangian (CEL) simulation of a standard experimental test and compare the model's predictive capabilities in both Lagrangian and Eulerian frameworks.

An important note is that only global mechanical response has been compared against experimental data. However, the local deformation gradients (nodal displacements) have not been compared to any experimental characterization. Thus, care must be taken when interpreting stress magnitude values calculated from the different tear simulations due to potential differences in the cross sectional areas created by nodal displacements at the maximum stress locations. Such experimental characterization requires significant additional work and the necessity of such data for constitutive model validation is dependent on the constitutive model's context of use.

5. Conclusions

The authors have discussed and illustrated efforts aimed at developing a material for highly deformable latex rubber that can be used in a wide variety of simulations. Implementations of the constitutive model in a tearing test not used to calibrate the model parameters resulted in good agreement with the experimental data. Additionally, the influence of boundary conditions on a specimen's mechanical response was clearly illustrated and reinforces the need to critically analyze simulation results for physical relevance and overall insight to the simulation's dominant physics.

6. References

1. Abaqus Analysis Manual, Version 6.14-1, Dassault Systémes Simulia Corp., Providence, RI.
2. Holzapfel, G.A., Nonlinear Solid Mechanics: A Continuum Approach For Engineering, John Wiley and Sons, Chichester, England, 2007.
3. PolyUMod, User's Manual, Version 2.6.2, Veryst Engineering, LLC, Needham Heights, MA.