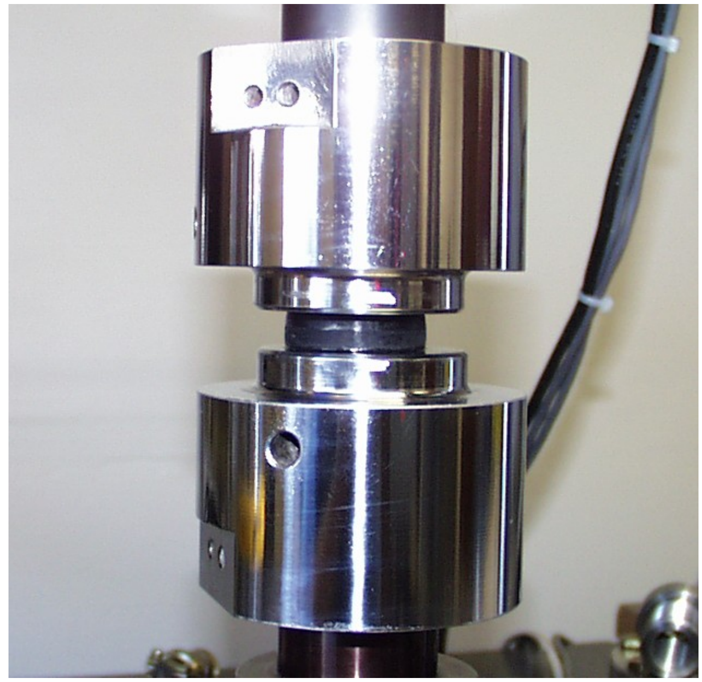
Material Model Calibration Using Abaqus and HEEDS

Summary

The ability of a finite element simulation to accurately capture the behavior of a structure strongly depends on the chosen material model. Not only must it be applicable to the given class of materials and intended application, it must be properly calibrated.

Sophisticated material models that use many parameters can present a challenging calibration task. Optimization techniques can be employed to determine suitable parameter values. In this Technology Brief, HEEDS, an optimization tool developed by Red Cedar Technology, will be used in conjunction with Abaqus to determine the parameters needed to model the viscoelastic behavior of a rate-dependent vinyl material.



Key Abaqus Features and Benefits

- Extensive catalog of material models, including hyperelasticity and viscoelasticity
- Ability to run Abaqus in conjunction with HEEDS to allow material parameter optimization

Background

Certain experimental tests, such as uniaxial tension or planar shear, produce homogeneous deformation modes that can be described analytically. The equations that describe the deformation can be used with the test data to calibrate a material model.

Calibration of a material model to match test data from a non-homogeneous deformation mode is more difficult, as an analytical representation is typically not available. One method that can be used in this case is optimization.

Elastomers can exhibit significant rate dependence and are often modeled using a combination of viscoelasticity and nonlinear elasticity. The material considered in this study is vinyl and the response will be simulated in Abaqus using the combination of a hyperelastic material to capture the quasi-static response, and a linear viscoelastic material to capture rate effects.

In this Technology Brief, the use of the HEEDS optimization software in conjunction with Abaqus is demonstrated for the calibration of the linear viscoelastic model.

Analysis Approach

The hyperelastic and viscoelastic material models are calibrated separately. The hyperelastic model parameters are determined exclusively with Abaqus, while the viscoelastic model is calibrated with Abaqus and HEEDS. The HEEDS optimization software package is used to automate the iterative execution of a sequence of Abaqus simulations. After each Abaqus analysis, the predicted response is compared to the experimental data. New values of the material parameters are determined via the HEEDS SHERPA algorithm. The Abaqus model is updated and executed again, and the cycle continues until the error between the predicted and experimentally measured responses becomes sufficiently small.

Hyperelastic Calibration

The hyperelastic model is calibrated in Abaqus/CAE with quasi-static tension and volumetric test data. Several strain energy potential forms are available in Abaqus; for the current study, the Marlow model is chosen for its ability to reproduce the test data exactly. The response of the Marlow model is shown in Fig. 1.

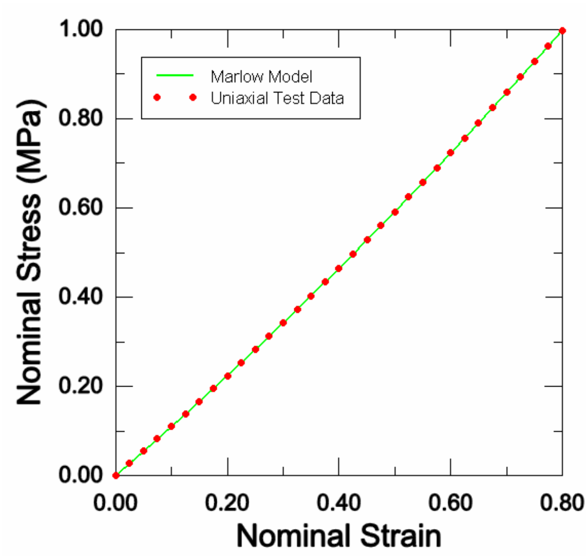


Fig 1: Comparison of Abaqus results with quasi-static uniaxial test data

Viscoelastic calibration

The linear viscoelastic portion of the material model is calibrated with bonded compression test data. The compression test is performed with the top and bottom surfaces of a cubical specimen bonded to rigid plates. As a result, the stress field is non homogenous with no analytical description. The test is performed at various rates, and strain rates of 10/s and 100/s are considered for the calibration process.

The test is simulated in Abaqus with a 1/8th symmetry model and a quasi-static solution procedure. A side view of the mesh is shown in Figure 2.

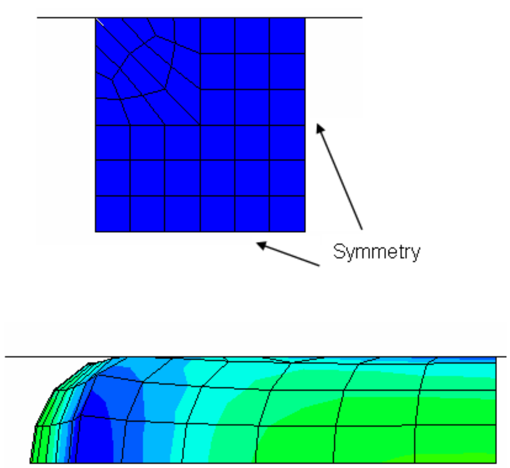


Fig 2: Abaqus finite element mesh, undeformed (top) and deformed (bottom)

For solid elastomers, a reasonable assumption can be made that there are no rate effects in the dilatational response; that is, we can set the volumetric Prony parameters k_i = 0. The other Prony series parameters g_i, τ_i are identified as the optimization parameters. For simplicity, a one term Prony series model is considered for the study, although any number of parameters may be considered.

The work flow for optimization is defined and automated in the HEEDS Modeler GUI. The optimization process is displayed graphically in Figure 4. A parametric script file drives Abaqus to run simulations at various loading rates.

The two parameters g_1, τ_1 are specified in an Abaqus input deck template and as the parameters are updated during the optimization, HEEDS creates a new input deck with the updated parameters and submits the corresponding Abaqus analysis.

The results from the simulation are compared to test data using a least-squares error computed with a Python script. A single cumulative least-squares value for all the different rate curves is calculated. The least-squares error is used in HEEDS as the objective function to minimize.

The results of the Abaqus analyses using the final values of g_1, τ_1 at the two loading rates are shown in Figure 3.

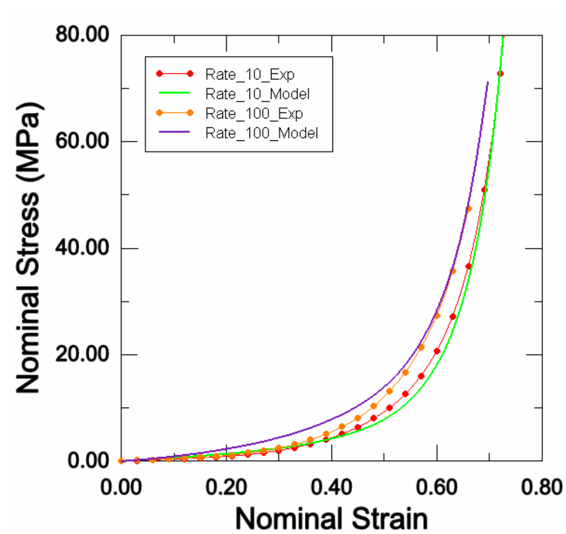


Fig 3: Comparison of bonded compression test data with Abaqus results

Conclusions

The HEEDS optimization software package can be used with Abaqus to determine material model parameters that may otherwise be difficult to calibrate. Successful calibration of a linear viscoelastic material model for the behavior of vinyl has been demonstrated.

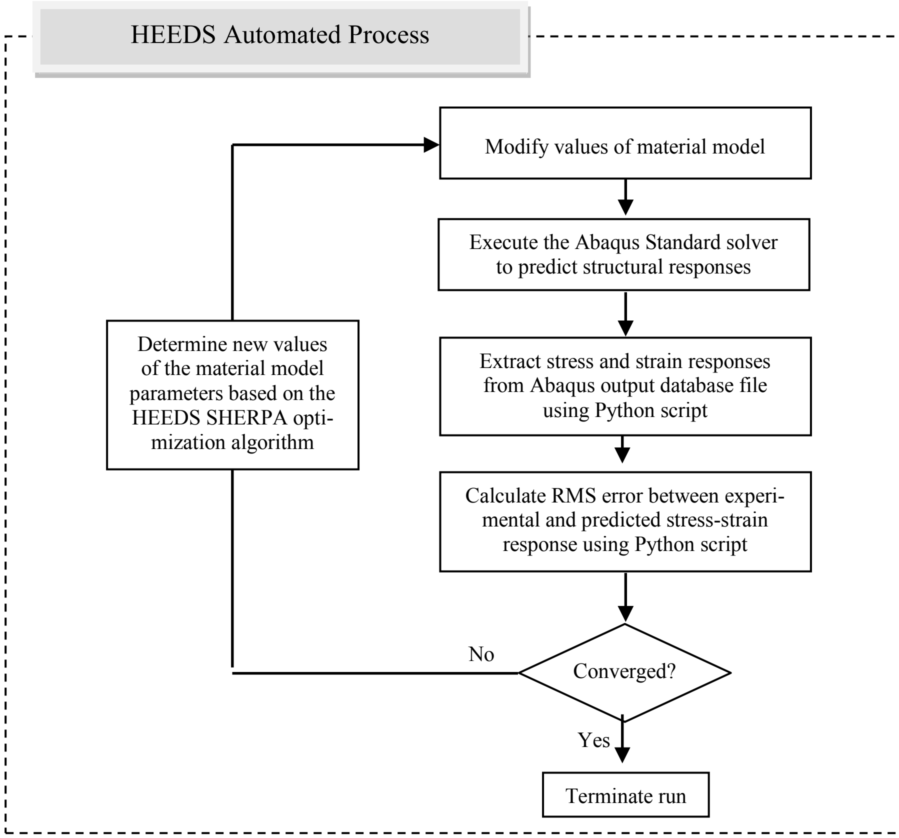


Fig 4: Automated Workflow for Optimization

Acknowledgements

The authors would like to thank Red Cedar Technology Inc. for providing valuable assistance during the course of this work.

References

1. *HEEDS User's Manual*, Red Cedar Technology, 2006.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief please see the following Abaqus 6.11 documentation references:

- Analysis User's Manual
  - "Hyperelastic behavior of rubberlike materials," Section 21.5.1
  - "Time domain viscoelasticity," Section 21.7.1