

Isight Optimization of Material Parameters used in the Abaqus Worldwide Side Impact Dummy

Summary

Finite element representations of crash test dummies are widely used in the simulation of vehicle safety systems. The biofidelity of such models is strongly dependent on the accurate representation of the nonlinear behavior of the constituent rubber, plastic, and foam materials.

Advanced material models are often needed to capture the dynamic response of the various parts of the dummy. The process of calibrating a finite element material model is resource intensive, as it involves optimizing model parameters to achieve good correlation with test data under different loading conditions and rates.

In this Technology Brief, an automated Isight optimization workflow for the calibration of material parameters is presented. Using component models from the Abaqus Worldwide Side Impact Dummy (WorldSID) we will validate one of the calibrated material parameters by comparing the response of several dummy sub-assemblies with experimental data.

Background

The WorldSID Anthropomorphic Test Device was developed by the WorldSID Task Group so that a globally harmonized side impact dummy would be available. Offering greater biofidelity than existing side impact dummies, the technologically advanced WorldSID will eventually replace the variety of dummies used in regulation and other testing [1].

Numerical simulation is a key part of the process of developing vehicle passive safety systems. It is thus essential that every hardware dummy have a virtual counterpart. During a vehicle program, hundreds of simulations are performed, and reliable dummy models provide large cost savings by reducing the need for physical prototyping and testing.

In 2006, Dassault Systèmes SIMULIA Corporation began the development of the Abaqus WorldSID 50th percentile (WorldSID50) model. The Abaqus WorldSID50 model has been developed in cooperation with the Partnership for Dummy Technology and Biomechanics (PDB), a consortium that includes German automobile manufacturers Audi, BMW, Daimler, Porsche, and Volkswagen. The consortium defined the requirements for the finite element dummy model and approved the release of the dummy for commercial use. The consortium also defined all material, component, and full dummy tests and requirements.



Key Abaqus Features and Benefits

- Automated simulation process workflows in Isight for easier identification of optimal material parameters, resulting in significant calibration time savings
- Extensive Abaqus material library includes rate sensitive foams, super-elastic alloys (Nitinol), hyperelastic, and viscoelastic representations

Finite Element Model and Analysis Approach

The Abaqus WorldSID50 mesh was provided by PDB and is based on CAD data published by ISO [1], technical drawings of the hardware WorldSID 50th percentile male model, and 3D scans of the dummy parts. The Abaqus WorldSID50 model consists of approximately 260,000 elements and more than 20 different materials (Figure 1) including rubber-like compounds, super-elastic alloys (Nitinol), foams, plastics, and vinyl.

The rate sensitive foam material model in Abaqus is used for the foams in the dummy. This model allows for direct input of stress-strain data at each strain rate for both tension and compression. An Abaqus user material that simulates super-elastic behavior is used to model both

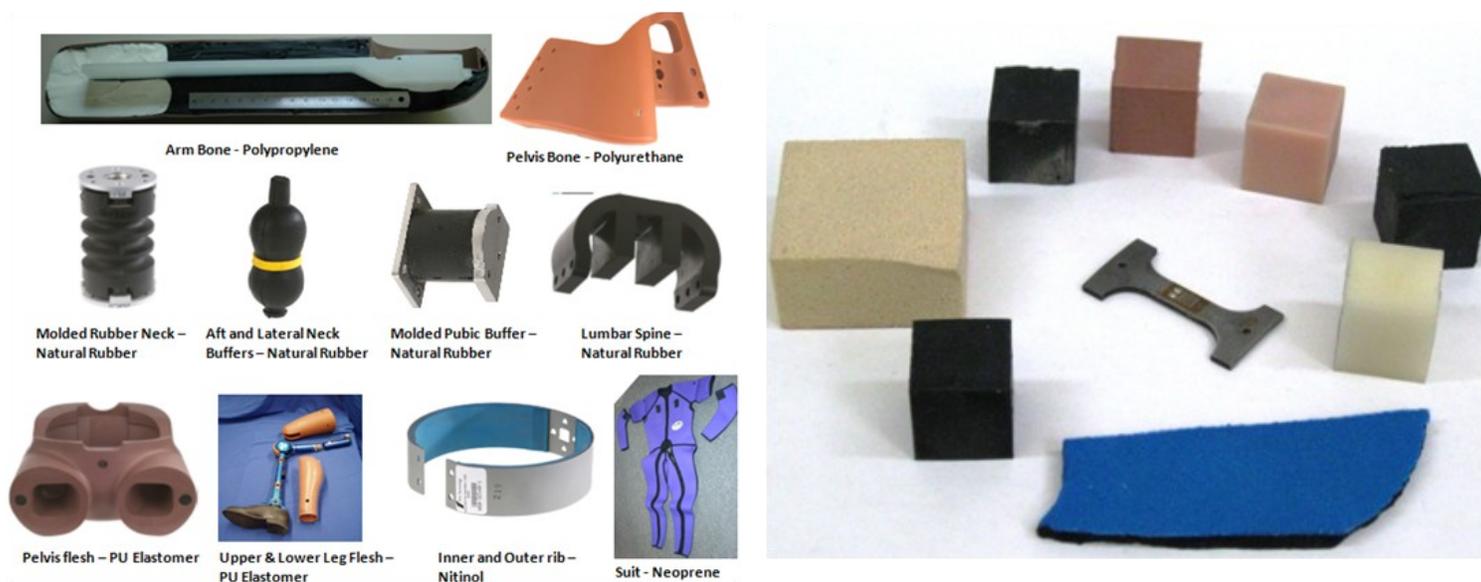


Figure 1: Components and materials of the WorldSID 50 dummy

the inner and outer Nitinol ribs. Hyperelastic material models are used in conjunction with a viscoelastic model (to account for strain rate effects) to simulate the rubber-like materials.

Most of the material samples used for testing were cut directly from the hardware dummy parts, and the rest were taken from material sheets. Experimental data were available in either nominal stress-strain format or pressure-compression ratio format (for volumetric tests). For each material, data from a variety of tests—quasi-static and dynamic (strain rates from 20/s to 400/s), tension and compression, volumetric compression, shear, and biaxial tension – were used for numerical material model calibration. Overall, data from over 400 material tests were used for the calibration.

Hyperelastic calibration

The constitutive behavior of a hyperelastic material is described in terms of a strain energy potential, which defines the strain energy stored in the material per unit of reference volume (volume in the initial configuration) as a function of the strain at that point in the material. Four strain energy potential forms were used in the finite element model: Arruda-Boyce, Marlow, Mooney-Rivlin, and the reduced polynomial form ($N=2$).

To help choose the most appropriate strain energy potential form for a given material, uniaxial and volumetric test data was directly input into the material calibration tool in Abaqus/CAE. For each strain energy potential, the response for each loading mode (uniaxial, biaxial, planar, and simple shear) was computed and plotted. In addition, the strain ranges for numerical stability of each potential were determined. Based on the stability check and a visual match with the test data, the appropriate hyperelastic material model was chosen.

Viscoelastic Calibration

The next step was to calibrate the strain rate effects in the hyperelastic material. The viscoelastic part of the material response was modeled using a Prony series expansion of the dimensionless relaxation modulus. The Prony series coefficients were calibrated using Isight.

Isight is a simulation process automation and design optimization software package that provides a suite of visual and flexible tools for creating simulation process flows (sim-flows) to automate the exploration of design alternatives and to identify optimal performance parameters.

An optimization sim-flow was constructed in Isight using the Prony terms as design variables. Different Abaqus analyses representing various loading rates both in tension and compression were used in the same Isight optimization sim-flow for calibration. The resulting simulation responses were matched against the test data using the “Data Matching” component of the Isight sim-flow.

Various statistical measures, such as sum of the squared difference, correlation factor, and area between the two curves, were used to quantify the deviation between the simulation and experimental curves. Various exploratory design space techniques as well as the Pointer technique were used in this multi-scenario, multi-objective optimization problem to find the best combination of Prony series coefficients that would minimize the difference between the simulation and experimental results for all loading types and rates considered.

The exploratory techniques (various types of Genetic Algorithms, Adaptive Simulated Annealing, etc) were selected because of their ability to cover the entire design space in an economical fashion while offering the best practical chance of identifying solutions within reasonable tolerance of the global optimum, even for highly non-

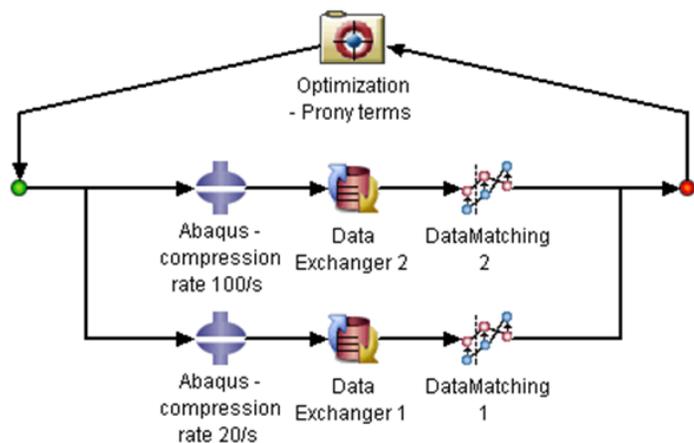


Figure 2: Optimization sim-flow for calibration of viscoelastic material parameters

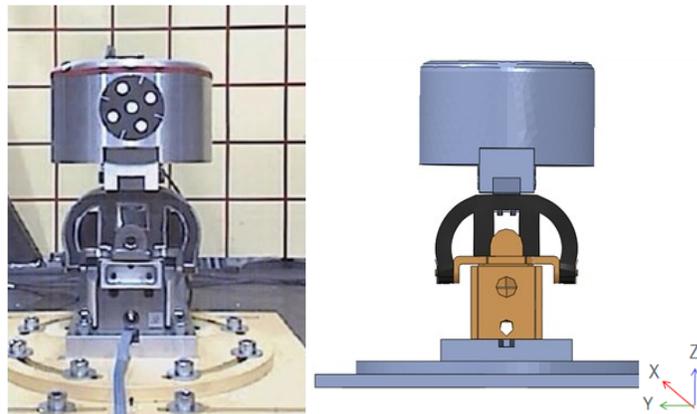


Figure 4: Lumbar spine component experimental test and model

linear design spaces. The Pointer technique uses a proprietary algorithm that allows it to switch between a set of complimentary optimization algorithms, enabling it to efficiently solve a wide range of problems in a fully-automatic manner.

Figure 2 shows the Isight optimization sim-flow used for the Prony series calibration for two loading rates. In Figure 3 we compare the simulated test results using the optimized properties against the experimental data for the material used in the molded rubber neck. The test specimens were subjected to dynamic compression with strain rates of 20/s and 100/s.

Results and Discussion

A variety of component-level experimental tests were simulated using the calibrated material models. These include sled-impulse tests (for neck and lumbar spine assemblies) and pendulum tests (for arm, iliac wings, and thorax ribs assemblies). Each test was repeated for various configurations of impact speed, impact location, impact angle, and impactor weight. The objective of the tests was to validate the calibrated material parameters and the subassembly models. All the component tests were designed by PDB to be representative of the loading conditions observed during a typical crash event.

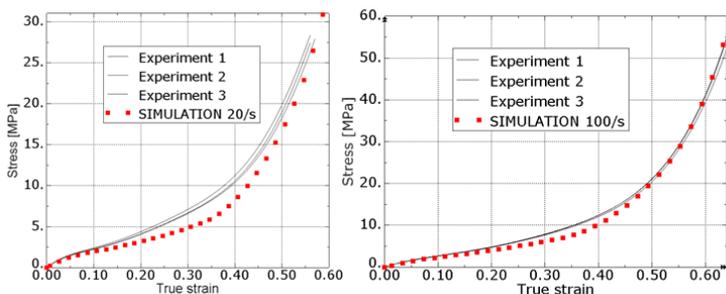


Figure 3: Dynamic compression test results and simulation using optimized Prony series parameters, molded neck material at 20/s and 100/s rates

Lumbar spine validation

As an example of the component testing, we consider the experiment and simulation of the lumbar spine assembly. The test configuration and model are shown in Figure 4. The lumbar spine is a critical component in the dummy as it plays a significant role in the upper body kinematics under lateral impact loads. Sled-type impulse tests were conducted for the validation of the lumbar spine.

The rubber lumbar spine is in an initial state of compression due to the weight of the thorax surrogate with a mass of 4.85 kg. The component is fixed at the bottom to a moving base. Sled acceleration pulses of 20g and 35g were applied separately in the lateral direction to simulate side impact. The tests were repeated for an angle of 90 degrees from the direction of travel (direct side impact) and at an angle of 60 degrees (oblique side impact).

Figure 5 shows a representative collection of output signals for one of the tested configurations: an impulse acceleration of 20 g applied to the base in the horizontal plane at a 60 degree angle with respect to the direction of travel. The output is measured by the lumbar spine load cell located at the center of the sacrum block (colored brown in Figure 4). The level of correlation between the simulation and experimental results clearly demonstrates that the finite element model of the lumbar spine using optimized material properties can realistically capture the shear and bending behavior induced by side impact loading.

Conclusions

Isight design automation and optimization software is a powerful tool for calibrating Abaqus material models. By automating the process of fitting material model coefficients to experimental data, time is saved and accuracy may be increased. Successful calibration of a linear viscoelastic material model for the behavior of rubber-like materials has been demonstrated.

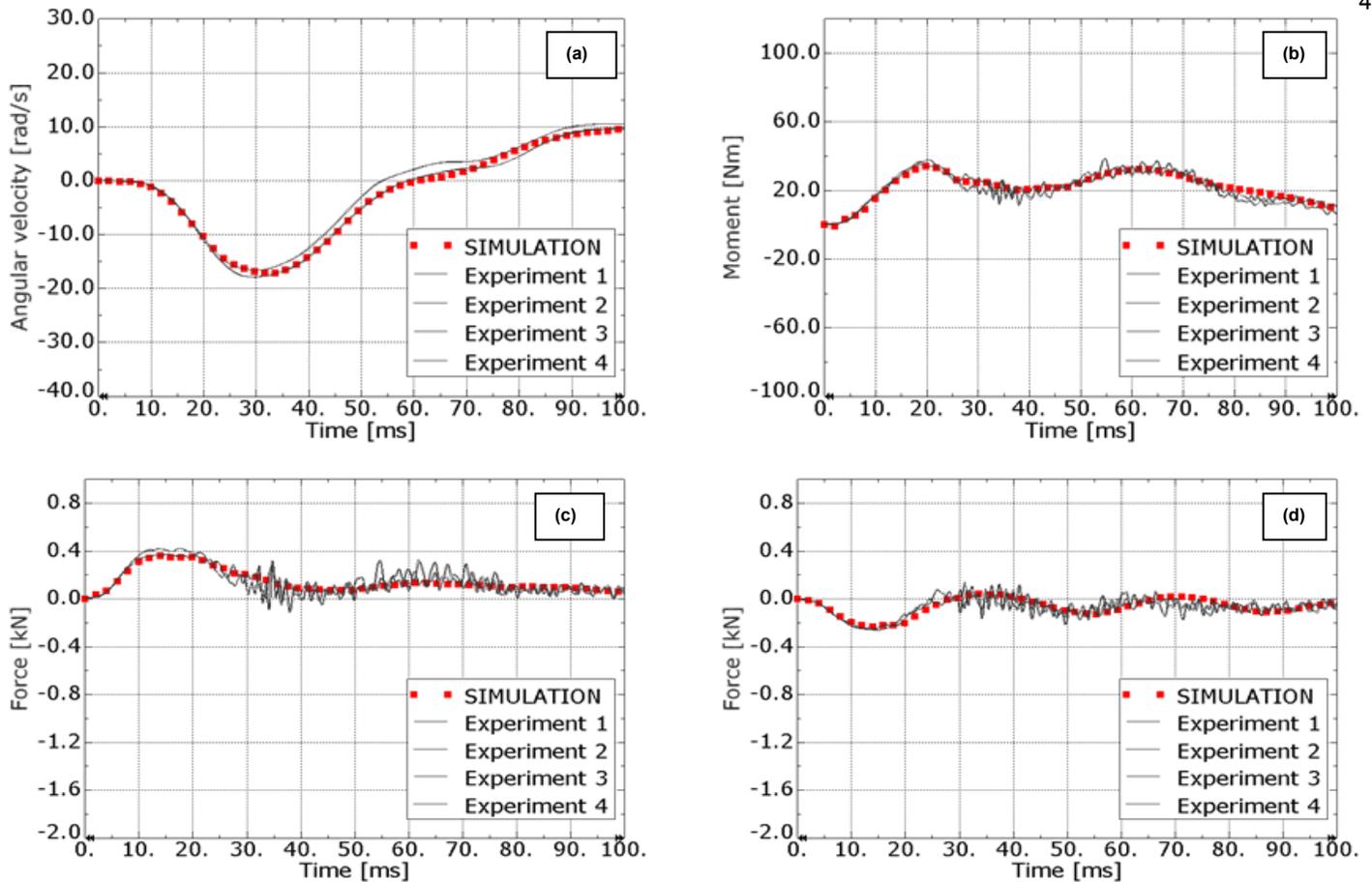


Figure 5: Lumbar spine test and simulation results – (a) Angular Velocity (VR_x); (b) Moment (M_x); (c) Y-direction force (F_y); and (d) X-direction force (F_x)

References

1. International Organization for Standardization, “International Standard 15830: Road Vehicles- Design and performance specifications for the WorldSID 50th percentile male side-impact dummy”, Geneva, 2005.

SIMULIA References

For additional information on the SIMULIA capabilities referred to in this brief please see the following documentation references.

- Abaqus 6.12 Analysis User’s Manual
 - “Hyperelastic behavior of rubberlike materials,” Section 22.5.1
 - “Time domain viscoelasticity”, Section 22.7.1
- Isight 5.6 Component Guide
 - Chapter 3. Using Application Components
 - Using the Abaqus Component
 - Using the Data Matching Component

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include Abaqus, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright Dassault Systèmes, 2011

