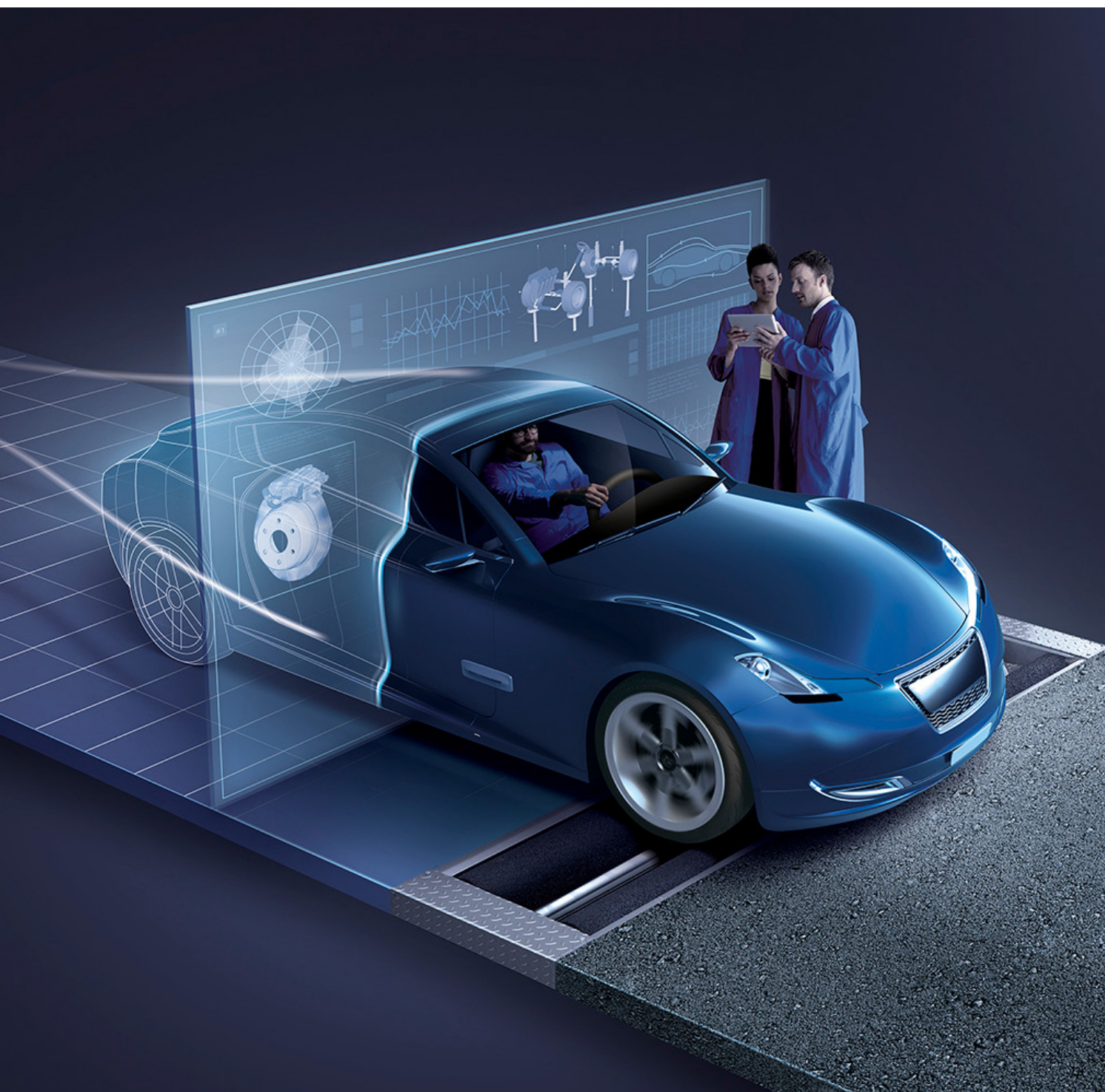


# DRIVING INNOVATION WITH REALISTIC SIMULATION

VOLUME 1



# Driving Innovation with Realistic Simulation

## Content

<b>Overview: Leveraging realistic simulation for global validation, proven performance</b>	3
<b>Mazda balances performance and weight in a steel car body</b> Isight streamlines and automates complex CAE optimization study	4
<b>Designing for a lighter future</b> Georg Fischer Automotive uses Tosca Structure optimization software to design lightweight components for improved fuel efficiency	6
<b>Ford Motor Company accelerates design for manufacturability of conical joints</b> Minimizing time and cost while meeting functional targets using Abaqus for CATIA and Isight	8
<b>Isight optimization of material parameters used in the Abaqus Worldwide Side Impact Dummy</b> SIMULIA Tech Brief: Simulation automation and optimization techniques help calibrate and validate material parameters	11

# Leveraging Realistic Simulation for Global Validation, Proven Performance

## Will your vehicles/components pass critical performance tests?

Progressively more sophisticated customer and industry demands have pushed vehicle performance and quality expectations ever higher. Simultaneously, advancing global competition requires the optimization of costs, acceleration of cycle time, and successful management of product quality, warranty claims and recall risks.

Global Validation, Proven Performance **3DEXPERIENCE®** solutions can enhance quality, save cycle time and reduce development costs by improving both your physical and virtual testing & validation. Reduce warranty claims and recall risks by leveraging holistic, integrated testing & validation capabilities, to ensure that all industry and customer requirements are successfully fulfilled.

## Optimizing vehicle performance

Early virtual simulation in the development process can save the costs, time & resources associated with physical testing on expensive prototypes later. The **3DEXPERIENCE®** platform integrates industry-leading realistic simulation and analysis capabilities, to validate and optimize design exploration, and extensive functionalities to address various workflows, multi-physics and open co-simulations.

Volume 1 of this eBook series provides case studies, papers, and tech briefs on how leading transportation OEMs and suppliers are leveraging realistic simulation and optimization applications to accelerate innovation and lower costs for evaluating, improving, and validating their vehicle/component performance:

**Manage Simulation Processes and Data**

**Balance Performance Requirements**

**Develop Lightweight Components**

**Save Time with Upfront Analysis**

**Meet Changing Market Needs**

**Evaluate Durability and Reliability**

**Calibrate Simulations with Test Results**



# Mazda Balances Performance and Weight in a Steel Car Body

Isight streamlines and automates complex CAE optimization study



Steel has remained the dominant material in car bodies for over a century by keeping pace with the evolving automobile. Improved corrosion-resistance, more refined mechanical properties, higher-strength characteristics, and advanced manufacturing technologies have kept steel at the top in terms of content in the average vehicle on the road—about 60% by weight today.

Yet weight remains a primary concern for automotive companies, due to its far-reaching effects on fuel efficiency. Other materials such as aluminum, magnesium, and composites are being increasingly considered as potential replacements for parts once made from steel. Although steel's proven reliability means it's likely to remain the primary ingredient in car bodies for some time to come, automobile manufacturers are now approaching the limits of how lightweight a steel car body can be. To fully understand and build within those limits, they are turning to sophisticated computer-aided engineering (CAE) tools that help them optimize their designs, provide the quality their customers demand, and meet ever more stringent mileage goals, emissions standards, and crash test regulations cost-effectively.

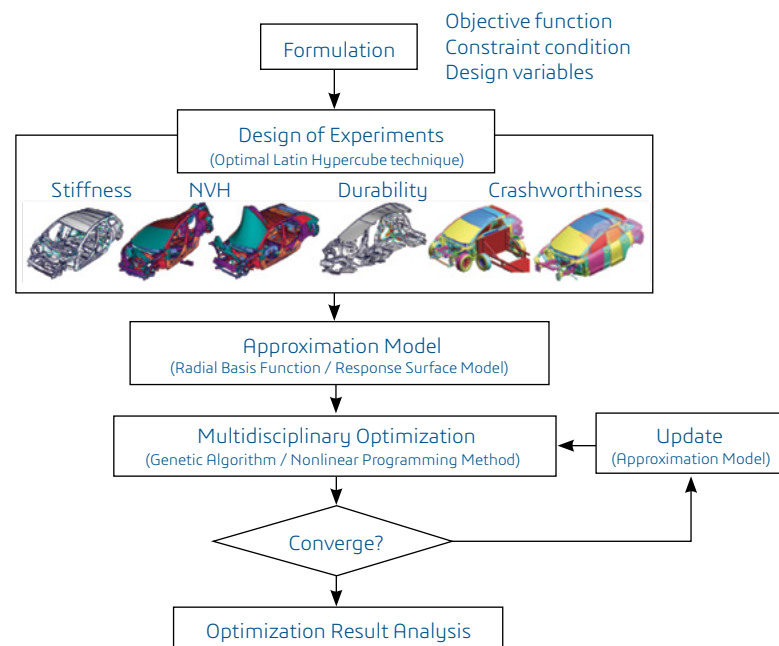
In a recent collaboration, the Vehicle Development Division and the Technical Research Center of Mazda Motor Corporation (Hiroshima City, Japan) developed a multidisciplinary methodology for design optimization (MDO) of a steel body structure based on the company's CX-5 car model.

"Optimization technology is essential for solving the problem of how to balance improved performance against reduction in weight," says Takehisa Kohira, technical specialist at Mazda.

The team's goal was to identify the lightest gauge (sheet metal thickness) combination of steel parts that would allow them to reach four target performance values—stiffness, NVH quality, durability, and crashworthiness—for the top-hat structure of the CX-5. The group employed a variety of CAE tools to model such diversity of the whole-car-body behavior—including Abaqus, LS-DYNA, Nastran, and others. "CAE helps us improve our designs, increase the accuracy of our analyses, and build fewer prototypes," says Kohira. Abaqus finite element analysis (FEA) evaluated strength, durability of body components, and thermal stress of powertrain components in vehicle development.

Creating an overarching analysis system that would first optimize vehicle body behavior in each of the four target areas, and then identify a final design that brought all these 'best' characteristics together at the lightest possible weight, was a complex analysis challenge.

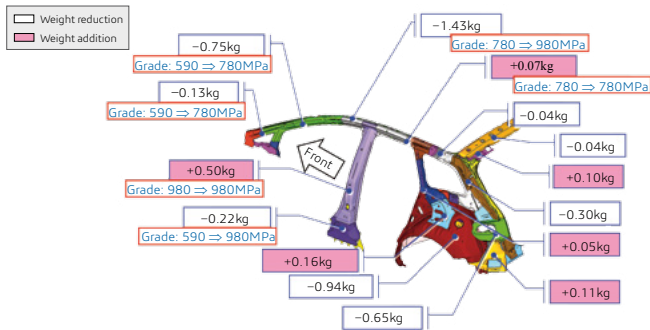
Among the target behaviors being examined, stiffness—both static and dynamic—involved primarily linear calculations. NVH analysis, on the other hand, entailed complex multi-physics problems that considered both the physical interaction of frame components and whole-body vibration. Crashworthiness,



Multidisciplinary Design Optimization (MDO) flow chart demonstrates the complexity of the challenges involved in taking weight out of a car design, while retaining a number of key performance characteristics.

Source: SIMULIA Community News





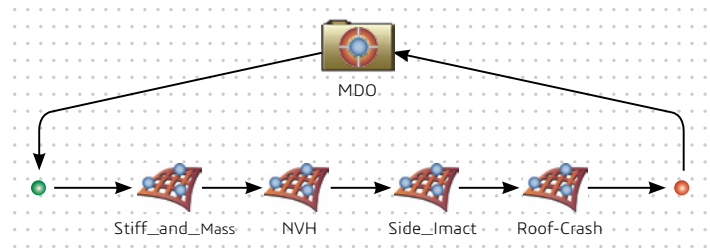
Initial analysis image of the portion of Mazda CX-5 body side frame assembly showing weight reduction (white labels) and weight addition (pink labels) that resulted from optimizing performance over a number of behavior parameters.

which is nonlinear to different degrees depending on whether it is front, rear, or side impact, presented the most complexity. "Side-crash analysis concerns only the bending/buckling domain, which can be predicted with an approximation model," says Kohira. "However, front- and rear-crash involve strong nonlinearity due to both buckling and axial compression of a multitude of parts, so the optimization of weight versus performance was particularly complex for these analyses."

To bring all the data together into a 'best-performance' body structure design, the team employed a variety of Design of Experiments (DOE) techniques and approximation models, manually conducting tradeoffs between the different behaviors. The car body's various performance targets were considered as constraints, and the design variables were thickness of material of each body component. "Our end goal was always the minimization of body weight," says Kohira. "But working to achieve that goal through manual data organization and comparison was taking a great deal of time."

Then the team turned to Isight for process automation and design exploration. "Once we started using Isight, we could more readily understand the limitations of our designs after our DOE studies, which made it easier for us to make decisions," says Kohira. "We could see the design space more clearly and better visualize our results."

Isight helped the engineers integrate all their CAE software into customized, 'drag and drop' workflows that would run all their performance tradeoff sequences automatically. "By



Isight process automation and design optimization software enabled Mazda to set up their MDO challenge within an automated workflow that reduced analysis setup and runtime considerably.

using Isight, we could confirm and numerically validate our designers' ideas and be confident in the validity of our designs," says Kohira.

An interesting effect of applying Isight to the car body optimization problem was the ability to categorize body components based on their importance relative to the performance of the whole structure. "During the optimization process, in most cases parts that had a low contribution to performance became thinner," says Kohira. "On the other hand, some of the parts that had a large contribution to performance needed to become thicker and heavier. Optimization with Isight enabled us to balance out these opposing needs while still lightening the overall weight."

The result? The team achieved their goal of a 3.4 percent reduction in weight over the previous design of the CX-5. Their multidisciplinary design optimization protocol is now being used in Mazda's SKYACTIV-BODY technology development program, which is aimed at improving vehicle fuel efficiency through engine and transmission development along with lightweight bodies and chassis.

Going forward, the Mazda team plans to adapt their Isight-automated MDO system to aluminum, CFRP (carbon fiber reinforced plastic), and other materials. "We have refined our steel designs about as far as they can go at this point," says Kohira. "Future designs will incorporate increasing proportions of a number of materials in addition to steel, but we now have the technology in hand to manage even greater complexity."

## Designing for a Lighter Future

**Georg Fischer Automotive uses Tosca Structure optimization software to design lightweight components for improved fuel efficiency**



As the auto industry worldwide works hard to improve fuel efficiency and meet increasingly stricter CO<sub>2</sub> emission standards, almost every conceivable way to make vehicles lighter in weight is on the digital drawing board of automotive engineers.

Such is certainly the case at Georg Fischer Automotive AG, the Swiss-based manufacturer and developer of high-performance casting components for the industry. As a key supplier that casts more than 600,000 metric tons of iron, aluminum, and magnesium into 100 million components yearly, the company is addressing weight through advances in designs, materials, and processes.

The benefits of such lightweighting strategies are clear and align with the company's "sustainable mobility" mission. A U.S. Department of Energy report says that reducing vehicle weight by 10 percent can increase fuel economy between six to eight percent.

### Designing the weight out with SIMULIA Tosca Structure

For GF Automotive, an essential part of the development of chassis components is the use of modern simulation and optimization software methods. Engineers at the firm have employed these computer-aided engineering (CAE) tools for years, achieving significant savings in product development time and upgrades in quality. The company currently relies on the optimization software suite, SIMULIA Tosca Structure,

with its topology and shape optimization modules, for reducing component weight while ensuring the product conforms to all required stress loads.

Reducing component weight requires a holistic approach to product development that combines engineering expertise with computer-based methods. Tosca™ Structure from FE-DESIGN (which has been part of the Dassault Systèmes **3DEXPERIENCE** technology portfolio under the SIMULIA brand since May, 2013) is a flexible, modular software system for non-parametric structural optimization. Its methods include topology, shape, and bead optimization that couple with industry standard finite element analysis solvers. GF Automotive uses the software on a wide variety of design projects in an effort to save weight from vehicle designs.

### Optimizing a steering knuckle: As light as possible—yet safe

In an automotive suspension system, the steering knuckle serves to attach the wheel to other suspension components (Figure 1). GF Automotive engineers—looking to eliminate weight from this component, while keeping the structure strong and safe—performed an optimization exercise using Tosca™ Structure software.

"Through the use of Tosca we get designs best adapted to loading, for lightweight and notch stress-free components in iron and light metal castings," said Roman Brauner, former project engineer in product development at GF Automotive.

The essential starting point for a successful high-volume (or series) design is to first define the appropriate design space. For the steering knuckle this included consideration of the axle, the wheel, and the kinematics of the suspension assembly. The component's product specifications were then transformed into mechanical requirements for the analysis.

Optimization to reduce weight began with the Tosca Structure topology module. The optimization created a lightweight, stiff

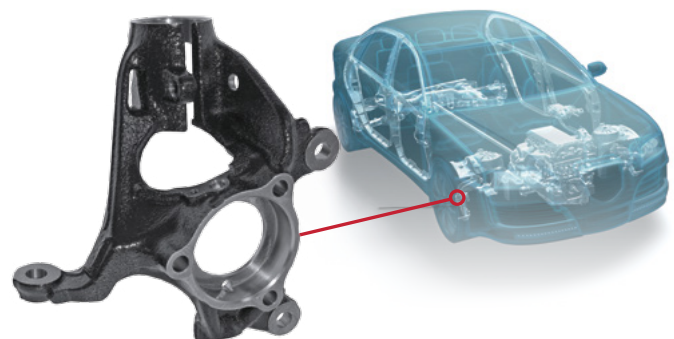


Figure 1. GF Automotive optimized steering knuckle (left) and location in wheel suspension system (right)

Source: SIMULIA Community News

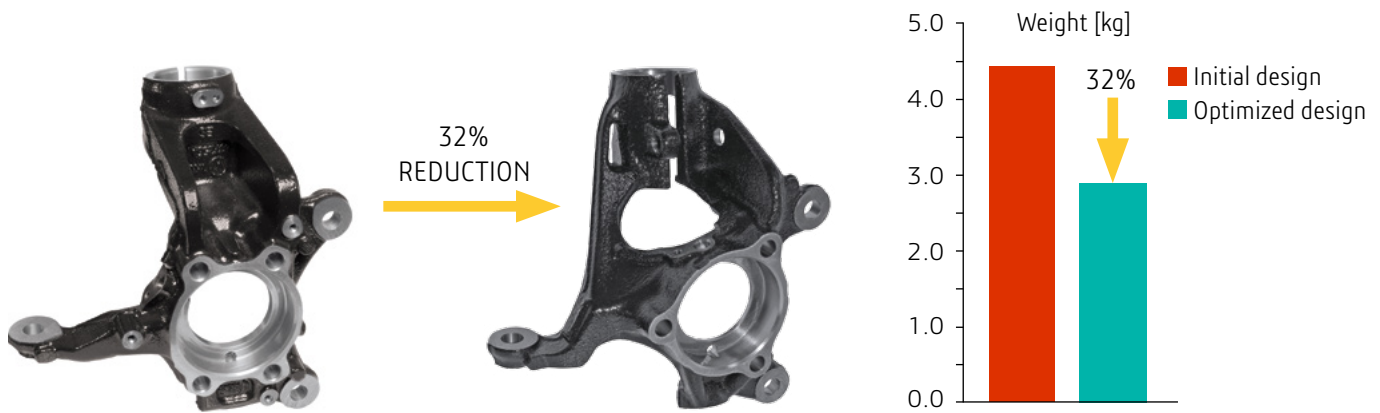


Figure 2. Tosca optimization of a steering knuckle. Optimized design with 32% weight reduction.

and strong, design concept considering all relevant operating load and load-abuse cases simultaneously—which is especially important for a dynamically stressed component such as a steering knuckle. The software’s shape optimization module was next used to eliminate an additional 100 grams of weight through modifications to the part’s surface geometry.

### Considering manufacturability and material choices for mass production

For a supplier that relies heavily on casting such as GF Automotive, topology and shape optimization are perfectly suited to their manufacturing methods because of the inherent design freedom of the process. Casting’s flexibility allows sophisticated optimized lightweight designs to be mass produced cost effectively and with high quality.

The benefits of the optimization process are many. For one, the new design concept can be easily transferred from Tosca™ Structure to a CAD model with little added interpretation effort.

Tosca™ Structure’s fully automated optimization process also provides results quickly. In the case of the steering knuckle, the software enabled engineers to easily arrive at a manufacturable design by considering critical manufacturing processes and steps—such as de-molding directions and clamping points for mechanical finishing—right in the optimization setup.

Value can be further added by selectively combining the improved design with the right material and a revised manufacturing process. GF Automotive engineered a new type of cast iron, SiboDur, developed in-house. This material is characterized by better properties at break, tensile strength\*, and fatigue strength. Given these properties, the lighter optimized design geometry complies with all safety requirements for the part.

“In addition to aluminum, innovative cast-iron materials are the trend,” said Brauner. “In each case, the aim is to use only the minimum amount of material required for the safe function of the component.”

### 32% lighter component yields reduced CO<sub>2</sub> emissions and wins an award

Following optimization results (Figure 2), the steering knuckle, which initially weighed 4.39 kg (9.7 lbs), is now 2.98 kg (6.6 lbs)—a 32 percent savings. Since each chassis requires two components, the total reduction is 2.82 kg (6.2 lbs) per vehicle. When looking at projected installation in 1.6 million vehicles, weight savings from this component alone would cut an estimated 11,600 tonnes (12,787 tons) of CO<sub>2</sub> emissions over the life of the part.

The new lighter steering knuckle has been implemented in large-series production since 2012. It’s being used by VW in their MQB platform, e.g. in the Golf VII and in the Audi A3, among others. The design has also received the prestigious environmental ÖkoGlobe, awarding pioneering innovations for sustainable mobility, in the category of “raw materials, materials, and process optimization.”

\*Tensile strength: The resistance of a material to a force tending to tear it apart, measured as the max tension the material can withstand without tearing.

Fatigue strength: The maximum stress a material can endure for a given number of stress cycles without breaking.

### GF Automotive AG—Passion for a lighter future:

The Georg Fischer Automotive AG, a division of Georg Fischer AG, with headquarter in Schaffhausen is a recognized development and serial production partner of the automotive industry with 10 production sites in three countries (Germany, Austria, China). The core business is the development and production of highly claimable castings of iron, aluminum and magnesium for the automotive industry and its suppliers. GF Automotive has therefore designed the research & development for years on weight reduction and lightweight and the reduction of CO<sub>2</sub> emissions and efficient fuel consumption.

Source: SIMULIA Community News

## Ford Motor Company Accelerates Design for Manufacturability of Conical Joints Using Abaqus for 3DS CATIA and Isight



Developing high-quality bolted joints is an integral part of vehicle chassis design. While less understood than the design of connecting members, such as a toe-link that connects the sub frame to the knuckle, robust joints are critical to improving handling and longevity of vehicle performance. Joints that are loose tend to exacerbate quality issues such as alignment, and ultimately the durability of the joined components. A properly designed joint is more efficient and can support larger loads with smaller size fasteners without loosening.

Engineers at Ford Motor Company were tasked to deliver a robust cantilevered conical joint design for the rear suspension system of a midsize passenger car (see Figure 1). To minimize time and cost while meeting functional targets, the team developed an automated Design of Experiments (DOE) process using Abaqus for CATIA (AFC) for structural analysis and Isight for process automation and optimization.

“Our team chose AFC in order to deploy standard stress modeling and simulation practices in the form of templates to a broader group of engineers within the design organization,” says Satyendra Savanur, chassis CAE engineer at Ford. “Linking Isight with AFC enabled us to develop a powerful and automated design analysis methodology. We used response surface model, one of the approximation models, for finding optimal parameters to size the joint.”

### Analyzing conical joint performance

A bolted joint is the most common type of attachment method used in the suspension of a car. In this application, a conical joint is used for connecting the toe-link to the rear knuckle with a cantilevered type connection. The two mating parts of the conical joint—the bushing inner sleeve and the knuckle—each have unique manufacturing tolerances of the cone angle.

To develop a robust conical joint between a steel inner sleeve and an aluminum knuckle the following aspects were considered: manufacturing tolerances of each component, contact area between the cone and seat, angle of the cone torque loss after the service load is removed.

To perform virtual tests of their design, the Ford engineers used AFC to create the finite element model of the knuckle and the bushing inner sleeve with the geometry input and material properties from their model created in CATIA. AFC maintains associativity with the CATIA model to ensure that the Abaqus model updates are robust when the CAD model is changed within the usable range of design variables.

During the physical assembly process, a forged steel inner cone is forced against an aluminum knuckle seat. Due to the different manufacturing processes used to make each part, the angular tolerances of the conical design features are different on the inner sleeve and the knuckle mating surface.

“Because of the potential angular mismatch, there are variations in contact area when the two surfaces mate together and the joint is fully torqued,” says Savanur. Local yielding can occur in the mating materials, leading to changes in contact area and pressure distribution during assembly of the joint. When the service load is applied, further changes to the contact area and contact pressure can occur.



Figure 1. Close-up view, before assembly, of the toe-link (black) and the rear knuckle (silver) using a conical joint.

Source: SIMULIA Community News



“It is, therefore, important to simulate both the joint assembly and the loading and unloading of service loads on the joint during the analysis,” he says. “Our objective was to deliver a robust conical joint design for the entire range of conical mismatch between the cone and the knuckle.”

For a robust contact analysis and even contact pressure distribution, the mesh of the inner sleeve was constructed to align with the mesh of the knuckle seat. To facilitate mesh alignment in the contact area, a separate “domain” of the knuckle seat (shown in turquoise in Figure 2) was created to simplify meshing. This part was connected to the rest of the knuckle body with a tied contact in Abaqus.

To simulate the bolt assembly process, a virtual bolt between the inner sleeve and the knuckle joint seat was created. External service loads were applied on the sleeve center. Nonlinear stress-strain curves for aluminum and steel were imported into AFC to facilitate the nonlinear analysis. Contact pairs and bolt tension were all created inside AFC. Output of contact area (CAREA) and contact force magnitude (CFNM) were possible using AFC for postprocessing. Finally, the Abaqus analysis file was output and submitted to the high-performance computing (HPC) cluster for running the analyses.

### Managing the DOE process

Ford’s need to evaluate a large number of designs with different combinations of parameters prompted the engineers to create an automated DOE process. In this process, CAD geometry updates and FEA model updates are completed in the same loop thus allowing a completely automated DOE approach.

At Ford, CATIA startup is customized with an external product management system. Scripting is used to strip away the linkages to the product management system before initializing the CATIA interface.

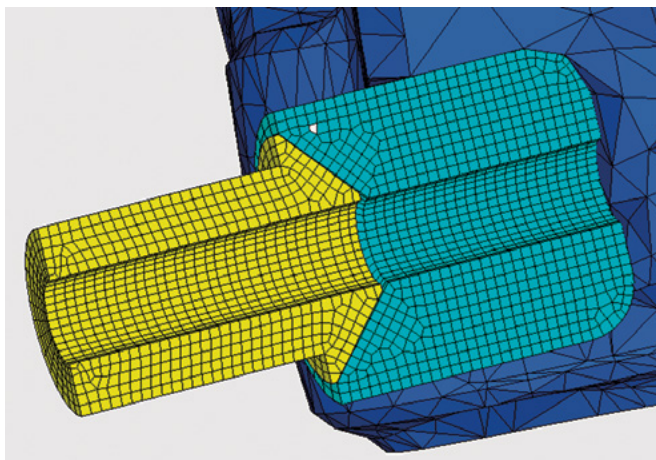


Figure 2. CAE mesh details of the conical joint.

Design parameters are then fed into CATIA with an external Excel file, a common method used to update a design table within CATIA. The input parameters from the Excel file are mapped to the DOE task of the Isight manager. This enabled automatic updates of the Excel sheet for each loop. Since Excel is synchronized with the design table, this results in automatic updates of the CAD geometry inside CATIA. Within AFC, geometry and FE mesh are associated, so the resulting mesh is updated to the changed CAD data.

“By developing a single integrated process, we were able to drive automatic updates of the geometry and mesh at the same time,” says Savanur. To manage and control the DOE process, Isight was used as the process automation manager. The resulting automation loop is completely integrated to run CATIA and AFC for CAD updates, create the Abaqus FE models, and submit job submission for analysis and post-process results.

The Abaqus component inside the Isight loop was used to extract outputs, including CAREA and CFNM for each run of the DOE (see Figure 3). The input parameters from the Excel file are then mapped to these output parameters to create an Isight approximation model.

“In our case, we used the response surface model method of approximation,” says Savanur. This approximate model of conical joint behavior can then be used to show how input affects output and quickly optimize the conical joint.

“This is the first application of an integrated DOE automation loop to morph geometry using CATIA with Abaqus at Ford,” says Savanur.

### Isight enables more efficient processes

The set-up and validation of the CATIA and AFC scripts, HPC job submission batch file, and the Windows batch command file took time and resources to develop, but were well worth it as they are reusable for subsequent projects with minor changes.

“Developing a comparable CATIA model with an associated Excel design table, and linked to an associated AFC model would take approximately three days to construct,” says Savanur. “Modifying and debugging the previously developed scripts to run with these new models would take another day. Using Isight, it took about 3.5 hours for the process to complete 35 analysis runs.”

“Typically, the manual CAE process consumes two days just to complete one run. Of course, this timing can be reduced if the project is critical, but this is the typical day-to-day turnaround time balancing several projects per engineer,” says Joe Peters, chassis CAE supervisor at Ford.

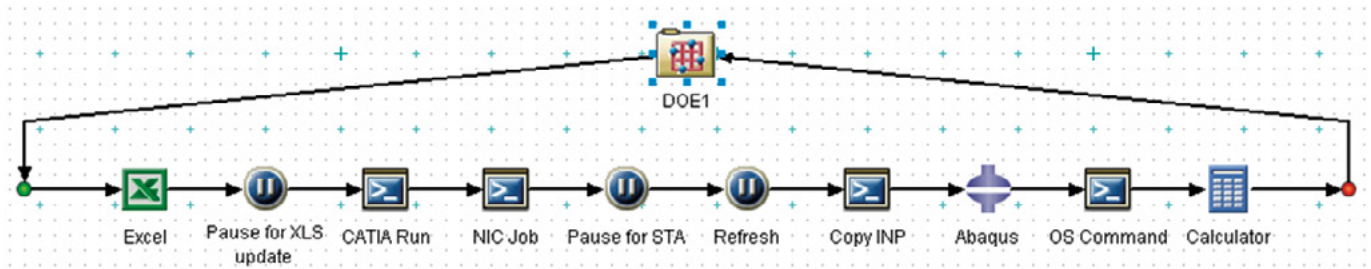


Figure 3. Integrated DOE automation loop using Isight.

Time inefficiencies typically occur in the transfer of data back and forth between CAE and CAD organizations, as people have multiple assignments and do not immediately stop their current work when new design iterations are requested; this is analogous to CPU time versus wall clock time.

"It is estimated it would have taken approximately 70 days to complete all 35 runs, while maintaining other day-to-day work; whereas, our new process eliminates the inefficiencies that were part of the manual CAD/CAE procedures," says Savanur. "By creating an integrated and automated closed-loop DOE process using Isight, we completed this task in about four days. This was the only way to help achieve the program objectives of cost and timing with a lean CAE organization."

"Using the automated DOE process, we were able to drastically cut down the time required to develop a robust conical joint with minimal resources," says Peters. "The largest amount of time savings was realized in the automated process of creating a CAE model from CAD. This is a testament to the fact that a small CAE team using new innovative technology helped Ford to achieve program objectives."

By using AFC and creating an integrated closed-loop DOE process with the help of Isight, Ford was able to deliver a robust conical joint design. This joint exhibits good contact area and retains clamp load after load removal, within the specified manufacturing tolerances.

# Isight Optimization of MaterialParameters 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.

## 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.

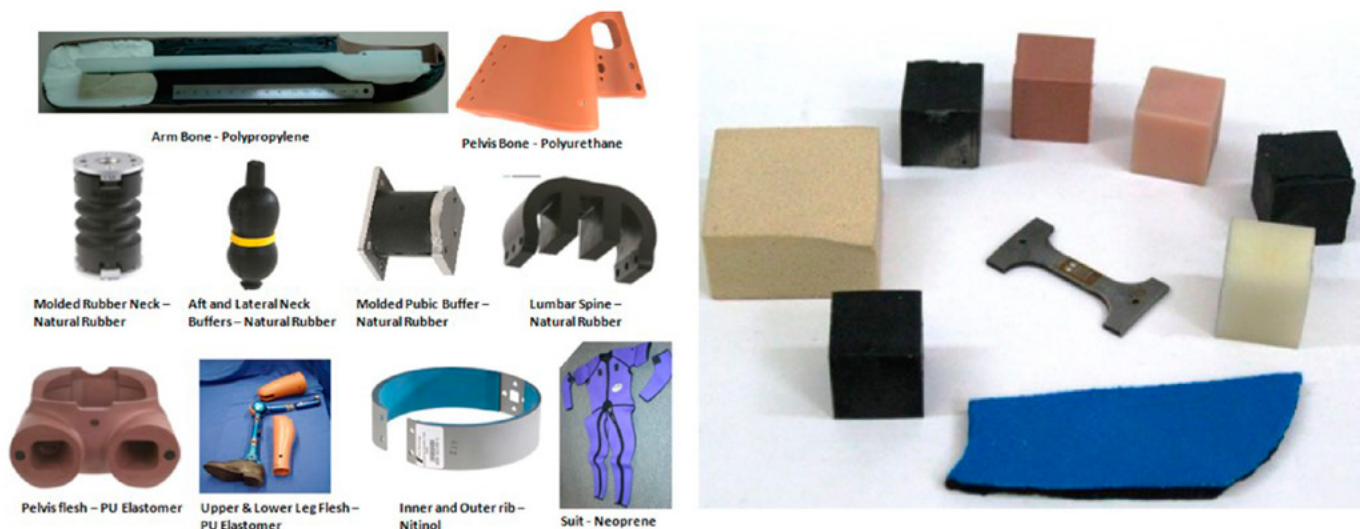


Figure 1: Components and materials of the WorldSID 50 dummy

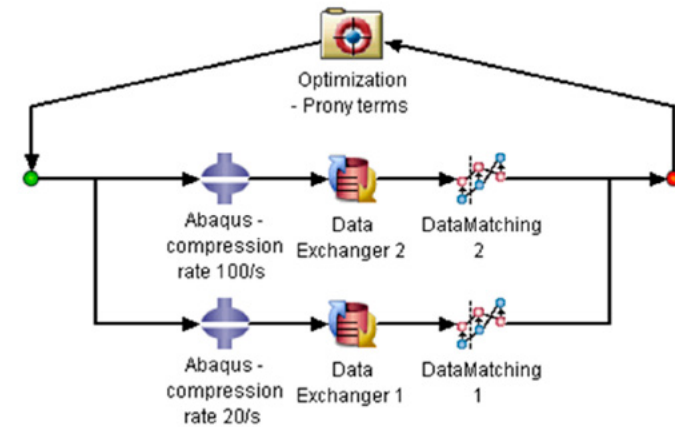


Figure 2: Optimization sim-flow for calibration of viscoelastic material parameters

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 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

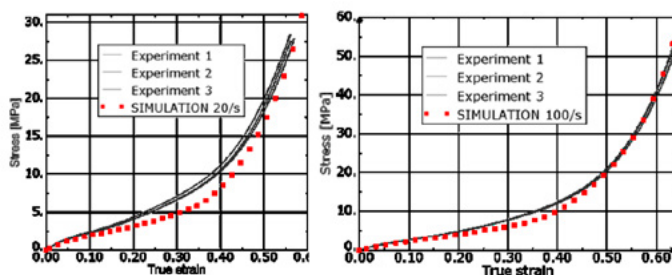


Figure 3: Dynamic compression test results and simulation using optimized Prony series parameters, molded neck material at 20/s and 100/s rates

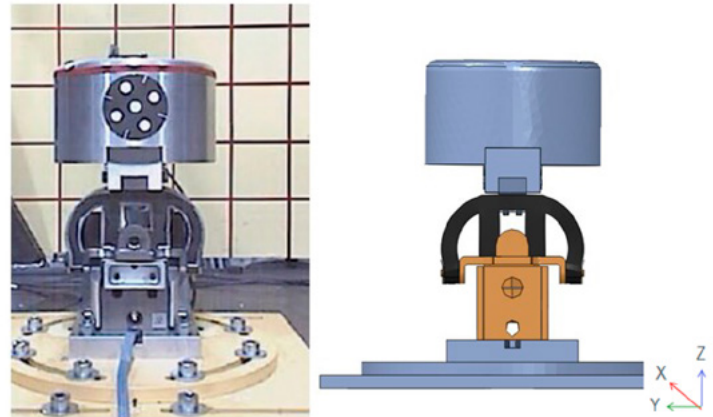


Figure 4: Lumbar spine component experimental test and model

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.



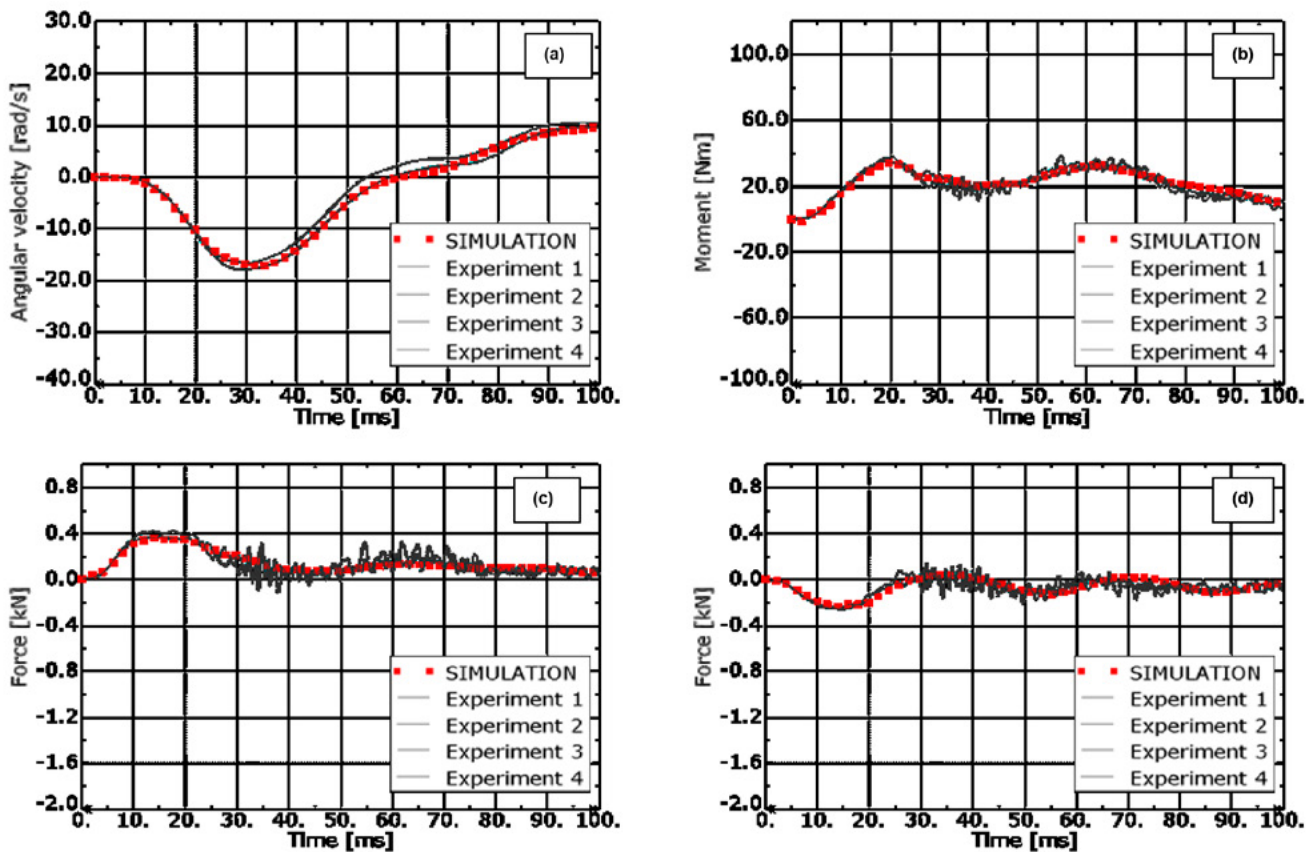


Figure 5: Lumbar spine test and simulation results – (a) Angular Velocity (VRx); (b) Moment (Mx); (c) Y-direction force (Fy); and (d) X-direction force (Fx)

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-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.

## 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.

## References

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 6.13 Analysis User's Guide:  
"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

---

## Our **3DEXPERIENCE®** platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3DEXPERIENCE®** Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 190,000 customers of all sizes in all industries in more than 140 countries. For more information, visit [www.3ds.com](http://www.3ds.com).



©2015 Dassault Systèmes. All rights reserved. **3DEXPERIENCE®**, the Compass icon and the 3DS logo, CATIA, SOLIDWORKS, ENOVIA, DELMIA, SIMULIA, GEODIA, 3DSWYDM, BIOVIA, NETVIBES, and 3DEXCITE are commercial trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the U.S. and/or other countries. All other trademarks are owned by their respective owners. Use of any Dassault Systèmes or its subsidiary trademarks is subject to their express written approval.