Development Methodology for a New Finite Element Model of the WorldSID 50th percentile Male Side Impact Dummy

George Scarlat[1], Sridhar Sankar[1]

Abstract This paper describes the modeling and validation process for the new WorldSID 50th percentile male dummy model (WorldSID) for the Abaqus finite element software suite. The objective of the project was to develop a robust and validated virtual dummy model that closely mimics the behavior of the hardware dummy in order to minimize the need for physical testing during vehicle design. Data obtained through extensive experimental tests were used to calibrate and validate the Abaqus model. The Abaqus WorldSID model shows good correlation with the hardware dummy model and was approved for use in production by the Partnership for Dummy Technology and Biomechanics.

Keywords Crashworthiness, Finite Elements, Numerical Simulation, Side Impact, WorldSID

I. INTRODUCTION

The hardware Worldwide Side Impact Dummy (WorldSID) Anthropomorphic Test Device (ATD) has been developed by the WorldSID Task Group with the purpose of having a technologically advanced side impact dummy which will have a greater biofidelity than the existing side impact dummies and which will eventually replace the variety of side impact dummies used in regulation and in other testing.

Numerical simulation, being an inherent part of the development process of the passive safety systems of vehicles, makes it essential that every hardware dummy have a virtual counterpart. Various numerical models of crash test dummies have been developed over the past two decades in major commercial finite element codes, ranging from rigid-body models to deformable models. The focus of the model development is continually shifting towards developing models of increased robustness that show a higher-level of correlation with the hardware. The German Association for Research in Automobile Technology (FAT) had established several working groups over a number of years which resulted in the development of high quality finite element models for US-SID, EuroSID 1, ES-2/ES-2re, and BioRID II.

Continuing with the same trend, numerical models for the WorldSID have been developed in various commercial finite element codes following the development of the hardware dummy. As more and more experience was gained, better and more comprehensive validation tests were designed and the quality of the subsequent finite element models improved. In late 2006, Dassault Systèmes Simulia Corporation started the development of the WorldSID finite element model for Abaqus to meet BMW's need for a virtual crash dummy model that closely mimics the behavior of the hardware WorldSID ATD. The Abaqus WorldSID model has been developed in cooperation with the Partnership for Dummy Technology and Biomechanics (PDB), a consortium that includes the following German automobile manufacturers: *Audi, BMW, Daimler, Porsche, and Volkswagen.*

II. METHODS

MODEL OVERVIEW – The Abaqus WorldSID model has 259,104 elements (Figure 1). The size of elements in the model ranges from 4 mm up to 15 mm and was chosen to ensure a stable time increment of approximately 1 microsecond for the explicit integration algorithm.

Certain advanced features in Abaqus have been invoked in the Abaqus WorldSID model in order to provide for accurate, yet efficient simulation of pertinent behaviors in the hardware dummy. Some of these advanced features are highlighted below:

Fig. 1. Abaqus WorldSID (w/o suit)

[1]) Dassault Systèmes SIMULIA Corp.

The rib cage assembly (Figure 2) includes a damping material intended to provide for a more realistic dynamic motion of the ribs when subjected to sudden impact loading in a crash event. Continuum shell elements (SC8R) are used to model this damping material; they have the geometry of three-dimensional solid elements, but with kinematic and constitutive behavior similar to shell elements. The three-dimensional element geometry allows for proper handling of the contact interactions that can develop in this region of the dummy, while the element kinematics and constitutive behavior essentially enable one element to be used to discretize the small thicknesses (4 mm to 7 mm) of the damping material.

The Abaqus WorldSID model uses connector elements (Figure 3) to reduce the size of the model by reducing the number of degrees of freedom. In the Abaqus WorldSID model, connector elements are used to model both the joints (hinges, ball-in-socket, etc) and the instrumentation (accelerometers, load cells, etc). Connector elements in Abaqus provide an easy and versatile way to model physical mechanisms whose geometry is discrete (i.e., node-to-node), yet the kinematic and kinetic relationships describing the connection are complex. The dummy has 14 joints modeled with 2-noded connector elements, which allow for pre-stress and friction behavior modeling. There are a total of 145 data channels in the model, including accelerometers, load cells, and infrared deflection measuring devices (IR-TRACCs) for Thorax ribs.

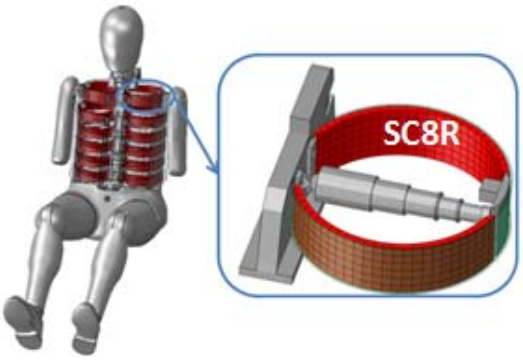

Fig. 2. Continuum shell elements

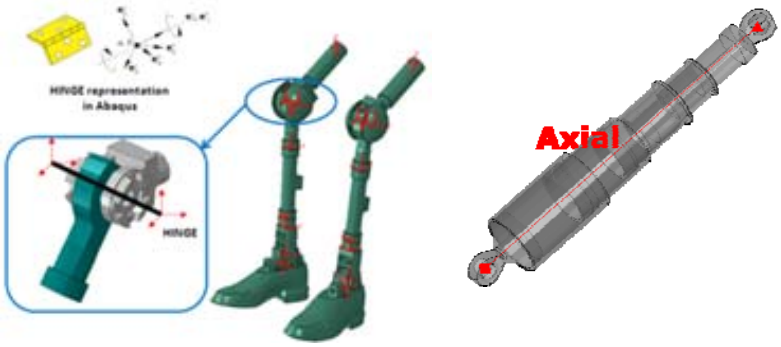

Fig. 3. Connector elements: Hinge (left) and Axial (right)

DEVELOPMENT PROCESS – A four step plan was followed for the development of the finite element model:

Version 0.0 of the finite element model contained the finite element mesh (assembled from the individually-meshed components of the model provided by PDB), correct masses and inertia, all joints, and all sensors. The finite element model was meshed based on CAD data published by ISO [1], technical drawings of the hardware WorldSID model, and 3D scans of the dummy parts. This release, which included generic material models, was used for pre-simulation studies to define the component-level tests for the validation work. Each sub-assembly (Head, Neck, Arms, Thorax, Shoulder, Abdomen, Pelvis, and Legs) was modeled and tested individually using both the implicit (Abaqus/Standard) and the explicit (Abaqus/Explicit) codes. This allowed for easy identification of all over-constraints and disjointed regions, thus increasing the model robustness. For instance, natural frequency extraction was performed on each subassembly to ensure that the parts were properly connected.

Version 1.0 included fully-calibrated material models as well as fully-validated component-level tests. Calibration here refers to determining the appropriate constitutive model in Abaqus for the experimental data measured from the material-level coupon-type tests. In total, 26 materials were calibrated and 124 component-level tests were validated. The calibrated materials in the Abaqus model include rubber-like materials, foams, plastics, vinyl, super-elastic alloy (Nitinol), and vibration damping material for the ribs. 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 calibration. A variety of component-level tests were simulated to evaluate the response of different sub-assemblies. These component-level tests include sled impulse tests (neck and lumbar spine) and pendulum tests (arm, iliac wings, and thorax ribs). Each of these tests was repeated for various configurations of impact speed, location and angle, as well as impactor weight.

Version 2.0 included full-dummy validation. Eight ATD certification tests targeting various regions of the dummy – head, neck, shoulder, thorax, abdomen and pelvis – and nine full-dummy side impact barrier tests with various combinations of dummy arm position, barrier shape, and impact speed were used to validate the finite element model. Version 3.0 will incorporate users' feedback on version 2.0 model based on use in their production environment.

III. RESULTS

MATERIAL CALIBRATION – Development of dummy models provides opportunities to invoke optimization methods in order to most efficiently arrive at the combination of model parameters that produce solution results to best match the experimental data. In particular, material model parameters can be optimized in an iterative and automated manner.

The WorldSID hardware includes many rubber-like materials, which are modeled in Abaqus as Hyperelastic materials with rate effects. To help choose the appropriate strain-energy potential model, uniaxial and volumetric test data were directly input into the material calibration tool in Abaqus/CAE. This tool evaluated different strain-energy potentials and plotted the response curves for each loading mode: uniaxial, biaxial, planar, and simple shear. In addition, the material calibration tool in Abaqus/CAE also assessed the numerical stability of the strain-energy potential models. Based on the stability check and a visual match with the test data, the appropriate Hyperelastic material model was chosen.

The rate effect was included in the material using viscoelasticity. The viscoelastic part of the material response was modeled using Prony series expansion coefficients of the dimensionless relaxation modulus, and the Prony coefficients were calibrated using Isight, an optimization software product of Dassault Systèmes SIMULIA. An optimization sim-flow (Figure 4) 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. The resulting simulation responses were matched against the test data using the "Data Matching" component of the Isight sim-flow, which provides a number of statistical measures to quantify the level of correlation between the simulation and experimental curves. Genetic algorithms based methods were used to find the optimal material parameters that yielded the best correlation.

Figures 5 and 6 show the comparison between simulation results and experimental results for a cubic sample model of the neck rubber material, which uses the optimized viscoelastic parameters, subjected to a dynamic compression test with strain rates of 20/s and 100/s respectively.

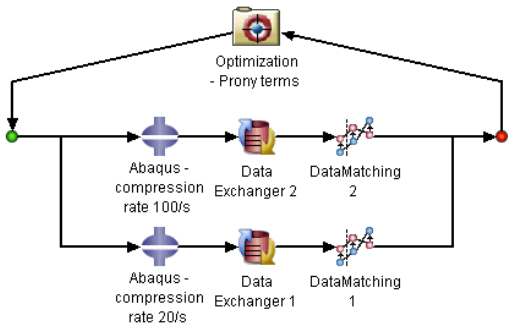

Fig. 4. Isight sim-flow for viscoelastic material parameters optimization

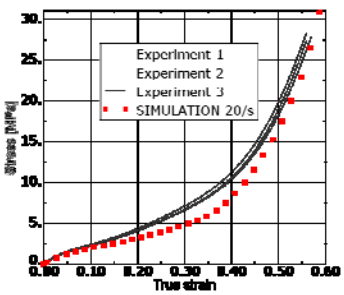

Fig. 5. Dynamic compression with strain rate 20/s

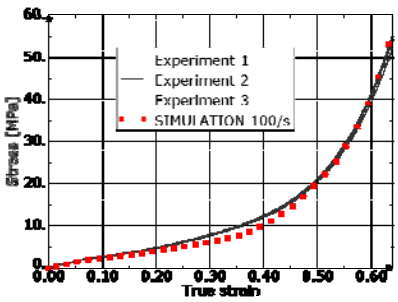

Fig. 6. Dynamic compression with strain rate 100/s

COMPONENT TESTS – Figure 7 shows the Lumbar Spine component test setup. 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. Figures 8 and 9 show representative output signals (simulation vs. experiment) for one of the tested configurations with an impulse acceleration of 20 g applied to the base in the horizontal plane, at a 60 degrees angle with respect to the lateral direction Y. All signals show good correlation with experiments, suggesting a good calibration of the lumbar spine component.

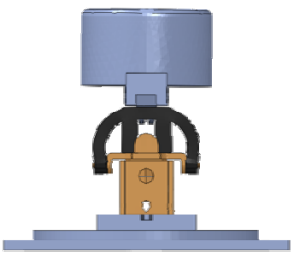

Fig. 7. Model setup

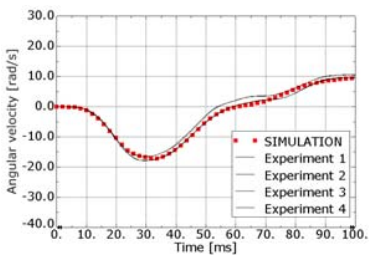

Fig. 8. Thorax angular velocity X

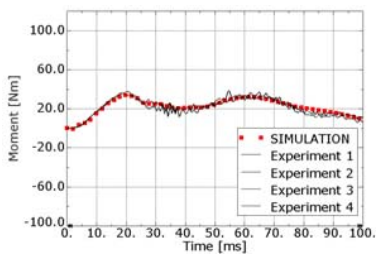

Fig. 9. Lumbar Spine moment X

CERTIFICATION TESTS – The ATD certification tests are intended to validate the hardware dummy prior to being shipped to the customers. All monitored signals from the hardware dummy must fall within specified corridors in order for the dummy to be considered fully certified. Similarly, the finite element model must pass those tests to be certified for production use.

Figures 10-15 show the thorax with arm certification test setup and comparisons of the experimental and simulation results. Similarly, Figures 16-18 show the pelvis certification test setup and the corresponding results. The certification test results show overall a good-level of correlation between the finite element model signals and signals from the hardware dummy.

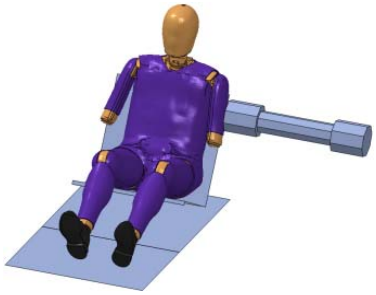

Fig. 10. Model setup

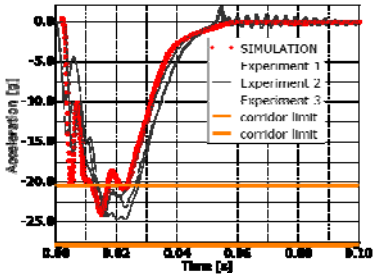

Fig. 11. Pendulum acceleration Y

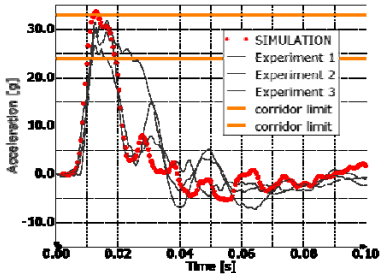

Fig. 12. Spine T4 acceleration Y

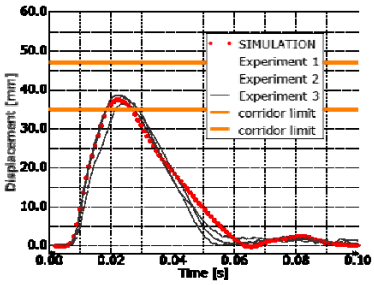

Fig. 13. Thorax rib 1 deflection

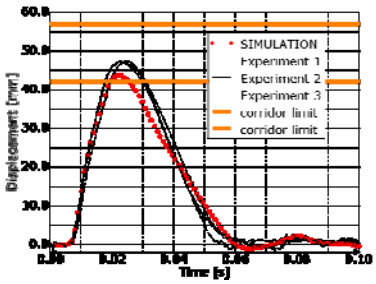

Fig. 14. Thorax rib 2 deflection

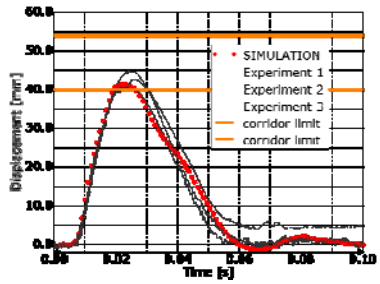

Fig. 15. Thorax rib 3 deflection

Fig. 16. Model setup

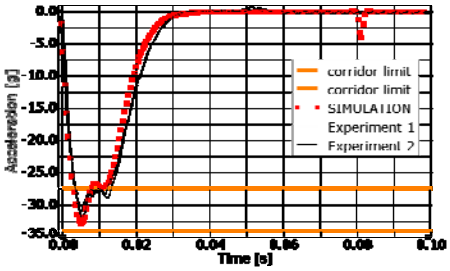

Fig. 17. Pendulum acceleration Y

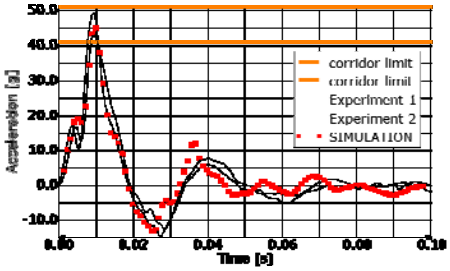

Fig. 18. Pelvis acceleration Y

IV. Discussion

The requirement from PDB for version 2.0 was to reproduce with a high level of accuracy both the dummy kinematics as well as all the relevant signals. For instance, in the full-dummy lateral barrier impact tests, the numerical model is required to show a good level of correlation for each of the 36 monitored signals (accelerations, forces, moments, and rib deflections) from the hardware dummy for all the 9 configurations tested. For the ATD certification tests, the finite element model response was required to fall within the corridors in order to be certified for production usage.

In an attempt to objectively assess the quality of the dummy model, statistical measures were used to quantify the level of correlation between the simulation results and the experimental results for the full-dummy lateral barrier impact tests. Out of the various validation metrics proposed in the literature for comparing transient responses, two measures – *Pearson correlation coefficient* and *Russell comprehensive error factor* – were selected for the purpose of the present study. The authors do not intend to propose that these are the best criteria for assessing the model quality, but rather these are merely useful measures to gain a quick insight into the model quality. Pearson product-moment correlation coefficient ("Pearson's r") is widely used in science as a measure of the strength of linear dependence between two variables. It can take values between -1 and 1,

with "-1" meaning the variables are completely anti-correlated, "0" meaning there is no linear correlation, and "1" meaning the variables are fully-correlated (i.e., both increase or decrease simultaneously). The Russell Comprehensive error factor (RC) [2] combines the error due to magnitude differences in the measured and simulated responses with the corresponding error due to phase differences. Small values of RC indicate a good agreement, while a large value indicates poorer agreement. As a rule of thumb, the suggested value of RC in order for the correlation to be considered acceptable is 0.2.

Table 1 shows an overview of the model quality as quantified by the two error indicators. A rather crude methodology was used to calculate the global values reported in the table. Individual values of the Pearson correlation coefficient and Russell comprehensive error factor were computed first to quantify the accuracy-level of each signal monitored in each of the full-dummy configurations tested. Since these metrics are unit-less, an average global value was then computed from these individual values. Furthermore, to gain additional insight into the quality of the dummy, average values were computed for certain dummy regions, such as Head and Neck, Thorax, etc. The obtained values suggest that the finite element model shows a good level of correlation for the loading cases considered.

Furthermore, using such approach, it is possible to gain some insight into how well various dummy regions mimic the corresponding hardware regions, enabling more targeted future development efforts. For instance, the results presented indicate that the accuracy of the Head-Neck region can be further improved.

Table 1. Statistics for model quality

Overall			
	Dummy		
		RC	0.125
		Pearson r	0.918
Per Region			
	Head-Neck		
		RC	0.145
		Pearson r	0.824
	Thorax		
		RC	0.118
		Pearson r	0.947
	Lumbar-Pelvis		
		RC	0.119
		Pearson r	0.952

V. CONCLUSIONS

The current Abaqus WorldSID model was validated for a variety of loading types and rates, which are intended to cover the range of expected loading conditions in a full-vehicle environment. However, it should be noted that the WorldSID 50th percentile is a relatively new anthropomorphic test device; consequently, the accumulated experience with this dummy is not very extensive. Hence, it may be possible that certain behavior is not covered in the finite element model even with the extensive database of validation tests used.

The Abaqus WorldSID model has been calibrated and validated based on more than 400 material, component and full-dummy tests. The finite element model shows good overall correlation with the physical dummy responses for the whole range of loading types and rates, successfully passing the stringent validation criteria set by PDB. The bottom-up methodology followed in validating the finite element model proved efficient, with minimal need for iterations between different levels (i.e. coupon, component and full-dummy) to reach the desired targets. The results obtained suggest that this model can be useful in predicting the dummy behavior in side-impact vehicle crash simulations. The feedback from PDB members and other customers using the current Abaqus WorldSID model will be incorporated in the next version of the Abaqus WorldSID model.

VI. ACKNOWLEDGEMENT

The Abaqus WorldSID finite element model development has been partially sponsored by the PDB consortium. The authors would like to thank the PDB group members for the experimental data provided and the numerous ideas shared, which helped to improve the model.

VII. 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.

[2] Russell, D.M., "Error Measures for Comparing Transient Data: Part I: Development of a Comprehensive Error Measure," Proceedings of the 68th Shock and Vibration Symposium, pages 175–184, 1997a