Migration of Crash Simulation Software at BMW

F. Dirschmid, H. Hooputra, H.U. Mader, H. Werner
BMW Group, Munich, Germany

Abstract: Over the past three years, BMW has undertaken a significant project to first investigate, then proceed with migration from its present crash simulation software to ABAQUS. This project has been motivated by a growing recognition within BMW that, in order to advance its crash simulation methods, improved robustness in the software employed, along with a greater emphasis on physically-motivated simulation capabilities, are required. This paper documents the processes that have been undertaken for the migration project, from the very early stages, up through to the present point where ABAQUS is being used exclusively in a substantial pilot project to design the crashworthiness aspects of a new BMW automobile. The paper also documents some ABAQUS advantages already becoming evident for crash simulation, along with some particular ABAQUS simulation results demonstrating its advanced capabilities for modeling failure mechanisms.

Keywords: Crashworthiness, Failure, Fracture, Impact, Occupant Modeling, Safety.

1. Introduction

BMW has used ABAQUS software for vehicle design analysis for several years, dating back to 1986, when material specialists started to use ABAQUS/Standard for thermo-mechanical analysis of the creep effects of particular engine components. Soon after this, engineers started to use ABAQUS for nonlinear static analysis of chassis components. In the 1990s numerous applications followed in the fields of chassis and engine analysis, such as the thermo-mechanical analysis of brake systems, including dynamic effects. Today, all nonlinear-static analysis work in the engine component, chassis, and body-and-white design are carried out in ABAQUS/Standard. In these applications ABAQUS has demonstrated particular fidelity in modeling material behavior and in capturing the important physical response of automotive components and systems, all the while providing robust solutions in a wide variety of applications. These factors are what motivated BMW to begin to investigate potentially using ABAQUS for crashworthiness simulation.
2. Preliminary Investigations

BMW initially approached ABAQUS (HKS) in 1998 to discuss the potential to develop next-generation crash simulation capabilities in the ABAQUS product suite. Recognizing the substantial efforts on both sides that would be required for success here, the parties agreed at that time to proceed deliberately first with a few trial projects.

The first trial project involved material modeling for crushable foams. The requirements for head impact injury criteria (FMVSS 201) are rather stringent, and energy absorption in the impact area relies heavily on crushable foams and similar materials. Hence, accurate modeling of these crushable foams is needed to obtain accurate results from head impact simulations.

BMW and ABAQUS agreed in 1999 to have a BMW employee pursuing a doctorate degree spend an extended period of three months at ABAQUS headquarters working with ABAQUS staff members to develop a new crushable foam material model for potential implementation in ABAQUS. These efforts first utilized the user material subroutine facility (VUMAT) in ABAQUS/Explicit; the model has since been implemented as a fully built-in material model (*CRUSHABLE FOAM) in both ABAQUS/Explicit and ABAQUS/Standard. BMW is now employing ABAQUS/Explicit for production headform impact simulation across several different vehicle programs, using the crushable foam model extensively [2, 3].

A similar trial project was initiated for modeling thermoplastic materials a year later, again involving an extended visit by a BMW employee to ABAQUS. This project served as a pilot to spearhead more extensive modeling efforts for thermoplastics now underway at both BMW and ABAQUS [4].

3. Migration Project

In August 2001 BMW and ABAQUS staff met to discuss establishing a more formal process to proceed with specifying and implementing crash simulation capabilities in ABAQUS. That meeting resulted in a document that outlined several key areas, including development focus, scheduling, milestones, extended staff engagements, technical support, and others.

It was agreed that development in ABAQUS focus first on structural crashworthiness capabilities, then transition to occupant safety considerations. A target vehicle was selected to further focus the development efforts. It was important for this vehicle to represent current crash simulation modeling practices employed at BMW; the vehicle chosen, the current S Series sedan (Figure 1), met these requirements. Significant milestones associated with specified crash loadcases (such as USNCAP) were subsequently established to ensure that ABAQUS was progressing satisfactorily in the evolution of its crash simulation capabilities.
Current full vehicle crash models at BMW are very large and complex, and can take substantial effort to get to run successfully in crash simulation software that has been in use for several years, even more in software that is undergoing significant developments in the area of crashworthiness.

To address this critical issue, it was agreed to carry out a joint effort between BMW and ABAQUS to catalog the existing full vehicle BMW 5 Series model, decomposing it into its various subsystems and components, then documenting each in sufficient detail. This was accomplished during a four-week visit by an ABAQUS staff member to BMW, working side-by-side with BMW engineers and consultants. This cataloging process then provided the basis for building up the equivalent car crash model in ABAQUS. Component, subsystem, and system models were built for ABAQUS using the decomposed models that resulted from the cataloging process. Each of these had to be run successfully at ABAQUS HQ before progressing to the next level. For example, body-in-white (BIW) impact against a rigid barrier had to run successfully before moving on to trimmed BIW against a rigid barrier. This systematic progression resulted in achieving a significant milestone on schedule: BMW 5 Series full vehicle model against a rigid barrier (USNCAP) in October 2002 [5].

This process also spawned a side benefit: the various component, subsystem, and system models provided the basis for specification and testing of a translator from BMW’s present crash software to ABAQUS.

Both BMW and ABAQUS recognized early that frequent communication would be necessary in order to keep the migration project on track. Weekly teleconferences were established in order to discuss and resolve various issues; these weekly teleconferences are still held today. Email correspondence, often on a daily basis, is also utilized to address and update technical support issues. In addition, several face-to-face status meetings have been held (eight meetings over the course of 3½ years). These meetings have provided an opportunity for detailed review of development progress, specification and scheduling of development projects, and in-depth technical exchanges.

The project team at BMW has grown steadily from the beginning of the migration project up to the present time, having currently about 15 people involved in the migration work, including managers, simulation specialists and consultants from all departments that use crash simulation software.

4. Pilot Production Project

With sufficient ABAQUS crashworthiness capabilities in place, and verified through testing of a range of crash loadcases with actual BMW models, BMW decided in 2003 to put ABAQUS through the rigors of production usage in the form of a pilot production project. This project is a new vehicle development program, and ABAQUS is being used exclusively for all crashworthiness simulations.
Production usage requires that ABAQUS fit in well with existing pre- and post-processing software utilized at BMW. BMW and ABAQUS have worked collaboratively with Altair Engineering (HyperWorks), GNS (Animator3), and S+C (scFEMod) to ensure that the ABAQUS software interfaces effectively with these pre- and post-processing tools. BMW’s specific needs and schedule requirements are established and communicated to each party.

In addition, BMW and ABAQUS have collaborated with First Technology Safety Systems (FTSS) to begin to develop a suite of finite element-based crash dummy models for ABAQUS. Occupant safety modeling requirements for the pilot production project have established the priorities and schedule for developing and delivering these models. Feedback provided by BMW on early usage of the dummy models has then been used to subsequently improve the fidelity and robustness of the models for production usage. A similar collaboration with Autoliv has been established to facilitate the development of airbag modeling capabilities in ABAQUS. Both BMW and Autoliv have prioritized the necessary airbag features to be implemented, have conducted early testing, and have provided feedback used to then refine and improve these features.

From the beginning, it has been planned to utilize ABAQUS development pre-releases for the pilot production project. This is in order to be able to take advantage of ongoing developments in ABAQUS, both in terms of new and enhanced features, as well as gains in performance. The schedule for pre-releases, along with the associated feature development, are set several months in advance. The risk of using development pre-releases has been mitigated by carrying out an extra measure of quality assurance before such pre-releases are delivered to BMW. A specific Crashworthiness QA test suite of problems has been mutually defined by BMW and ABAQUS to reflect the types of models to be used in the pilot project. The Crashworthiness QA test suite is then executed and evaluated at ABAQUS prior to shipping the pre-release to BMW. The contents of the Crashworthiness QA suite are also reviewed and updated periodically to ensure these problems reflect the current modeling practices employed at BMW.

The pilot production project has been running now since September 2004, using ABAQUS/Explicit exclusively for the crashworthiness design simulations. Initial work has focused on structural crashworthiness aspects for this new vehicle, using already large and complex models. With more than 500 simulation runs for all relevant load cases, ABAQUS/Explicit is demonstrating a strong degree of robustness in its application to the demands of a production crashworthiness program. Occupant safety simulations for the pilot production project have been started only recently with ABAQUS/Explicit, but will be ramped up to full service until mid 2005.

5. **Modeling of Failure Mechanisms**

BMW is constantly striving to improve the accuracy of its crashworthiness simulation methodologies. This involves active ongoing investigations in two primary areas that have a substantial impact on the accuracy of crashworthiness simulation: structural behavior and
response, and the behavior of occupant safety systems (airbags, seatbelts, interior trim, and the dummy models themselves).

Modern automobiles are incorporating more lightweight materials (such as aluminum) and advanced connection and joining techniques (such as structural adhesives). Overall structural response during a crash event can be significantly affected by the plasticity, damage, and failure mechanisms of these materials and connections. Hence, it is important to capture these mechanisms properly in order to accurately predict the overall structural response for crashworthiness simulation.

BMW has been working closely with the ABAQUS development staff, as well as other industry and research partners, to develop advanced failure modeling capabilities, including those for materials, point fasteners (spotwelds and rivets), and structural adhesives. For each, controlled physical tests are carried out to obtain experimental results, which are subsequently used to validate and calibrate the corresponding modeling functionality.

Figure 2 shows a double chamber column made of extruded aluminum subjected to quasi-static axial compression; Figure 3 shows the resulting deformations when the column is subjected to dynamic compressive loading conditions. The differences in deformation modes are clear. Under quasi-static load, the column buckles and folds upon itself in a regular pattern. Failure of the aluminum material occurs locally but is not dominating the global behavior. Under dynamic loads, the response is quite different, as severe tearing and fracture of the aluminum are evident. ABAQUS/Explicit simulation results for both loadcases are also shown in these figures, with good correlation observed for both scenarios.

A second example demonstrating a severe three-point bending test is shown in Figure 4. The experimental results show that the profile collapses underneath the punch, along with intensive folding of the upper edges of the profile. The localized straining on these edges causes rupture of the material. The corresponding ABAQUS/Explicit simulation exactly replicates the folding pattern, and also predicts well the regions of high failure potential (Figure 5, with failure potential mapped onto the undeformed mesh). The punch force-deflection response for this test is also plotted in Figure 6. Three samples were tested experimentally; the narrow bandwidth exhibited by the test results demonstrates good experimental repeatability. The predicted force-deflection response matches the experimental curve almost perfectly for higher punch displacements. The peak force level at the onset of punch displacement is about 5% lower to the experiment due to stiffening effects on the T-intersections of the double chamber profile that are not covered adequately by the used shell mesh.

Structural adhesives are used in body-in-white structures to increase the stiffness of spot-welded flanges. They also increase the impact strength of the components where they are applied; both the adhesive and the spotwelds contribute to holding the component flanges together and maintaining strength over a greater range of severe loading. However, under severe crash loading, the adhesive can fail, and thus its failure characteristics must be calibrated and subsequently modeled in a crashworthiness simulation.
ABAQUS presently provides a library of cohesive elements to model adhesive joints, including capabilities to model de-bonding failure mechanisms that can occur under severe loading. A continuum-based modeling of the adhesive is appropriate when the adhesive layer has a finite thickness. The macroscopic properties, such as stiffness and strength, of the adhesive material can be measured experimentally and used directly for modeling purposes.

Figure 7 shows a T-peel test configuration set up to measure the failure characteristics of a typical structural adhesive used in automotive applications. Due to the complexity of failure mechanics with adhesives and the restrictions of meshing in full-size crash models (element size not smaller than 5 mm), such a T-peel configuration is used to measure and calibrate the adhesive material parameters rather than attempt to do so with a pure material testing approach. Figure 8 shows a comparison of experimental force-time response versus that obtained from corresponding ABAQUS/Explicit simulations. An automated optimization process for a mesh size of 4x4 mm yields a set of parameters for plasticity, damage initiation and damage evolution that can replicate the experimental force-time curve very closely (red line). The ripples are discretization effects and decrease with reduced mesh size. For application in full-size car crash models it is even more important to see, that with a mesh size of 8x8 mm also a good correlation to testing can be achieved (gray line). Ongoing research work will show, if material parameters derived from basic test configurations are robust enough to be transferred to a wider range of geometries and loading conditions and finally to predict deformation in full vehicle crash simulation models.

6. Summary

Over the past three years, BMW and ABAQUS have worked closely to migrate BMW from its existing crashworthiness software to ABAQUS. While the level of effort required is substantial, the progress achieved thus far demonstrates the feasibility, given a strong commitment and close collaboration from both sides. ABAQUS/Explicit is presently being used exclusively for the crashworthiness design simulations in a new vehicle project, and is demonstrating a strong degree of robustness in meeting the demands of such a production program.

As the need to achieve a greater degree of accuracy and predictability in crashworthiness design simulations continues to expand, this places a corresponding requirement to improve the technology for modeling damage and failure mechanisms. Such mechanisms have a significant impact on the evolving structural response during a crash event. BMW and ABAQUS are working closely to define requirements for, and to implement appropriate capabilities in ABAQUS. Some of these features are already available in ABAQUS Version 6.5 with further work underway for Version 6.6; BMW is undertaking efforts now to calibrate these features for usage in the pilot production project already underway.
7. References

Erfahrungen mit ABAQUS/Explicit bei der Modellierung des crashrelevanten Verhaltens von Polyamerschaumstoffen
ABAQUS Anwendertreffen, Essen, Germany, October 1, 1999

Finite Element Analysis of Head Impact According to Extended FMVSS 201
ABAQUS Users’ Conference, Newport, USA, 2002

Simulation des Kopfaufprall nach FMVSS 201 mit ABAQUS/Explicit
VDI Tagung Berechnung und Simulation im Fahrzeugbau, VDI Berichte, Würzburg 2002

Characterization and Modeling of Thermoplastic Materials for Crash Applications
CrashMAT 2002, Freiburg, Germany, April 15-16, 2002

Crashworthiness Simulation of Automobiles with ABAQUS/Explicit
ABAQUS Users’ Conference, Munich, Germany, 2003
8. Figures

Figure 1. First car crash simulation model in ABAQUS/Explicit (BMW 5 Series sedan), load case EuroNCAP.
Figure 2. Quasi-static compression of a double-chamber extruded aluminum column. Left figure shows experimental results. Right figure shows corresponding ABAQUS/Explicit simulation results.

Figure 3. Dynamic compression of a double-chamber extruded aluminum column. Left figure shows experimental results. Right figure shows corresponding ABAQUS/Explicit simulation results.
**Figure 4.** Double-chamber aluminum profile subjected to severe three-point bending. Left figure shows experimental results. Right figure shows corresponding ABAQUS/Explicit simulation results.

**Figure 5.** Failure potential in double-chamber aluminum profile subjected to severe three-point bending. Red elements (plotted on undeformed mesh) indicate areas of failure.

**Figure 6.** Comparison of load-deflection response for severe three-point bending test.
Figure 7. T-peel specimen to measure failure characteristics for structural adhesives.

Figure 8. Comparison of force-time response for T-peel specimen.

Visit the SIMULIA Resource Center for more customer examples.