Crashworthiness Simulation of Automobiles with ABAQUS/Explicit

Touraj Gholami, Jürgen Lescheticky, Ralf Paßmann
BMW Group, Munich

Abstract
Passive safety simulation is a well established tool in the development process of automobiles at BMW Group. Based on the demands on crash analysis software, which has been defined in a crash vision 2004, ABAQUS/Explicit has been tested for crashworthiness simulation. The recent introductions of a general contact capability and a new spotweld modeling technique have been key to enabling this testing. First experiences of component-based applications and of whole car crash analysis exemplify the strong architecture and existing complement of robust features in ABAQUS/Explicit, and demonstrate its general feasibility for crash simulation.

Introduction
The increasing legal and customer demands on passive safety of automobiles have to be fulfilled under the conditions of shortened development times and cost reductions. Today the design process of a car with regard to its crashworthiness function is driven by a virtual development at BMW Group. A wide range of different applications has to be covered by simulation influencing the design of body in white, interior and exterior trim, chassis and powertrain. When global results of whole car crash simulation have been the primary focus the last years, because of the increasing reliance of the design processes on numerical results, a more local, detailed analysis is needed today. These demands on the local accuracy of the simulation results lead to a substantial increase of further development work of the numerical methods. But, high-end material models, increasing complexity of the used car models and complex connecting techniques have to be based on a finite element code of high quality and stability. A crash vision 2004 has been defined at BMW and finite element codes are tested for their reliability to meet the described demands. Within this paper a short overview about crashworthiness simulation of automobiles at BMW Group is given. An introduction into the crash vision and the associated software development are presented. The procedure to test ABAQUS/Explicit for these applications is explained. Several examples for testing basic functionality and general feasibility are shown.

Crashworthiness Simulation at BMW Group
The development process of automobiles with regard to crashworthiness behaviour depends strongly on virtual testing and simulation. Thus, development work based on cost intensive prototype construction and testing has been extensively reduced for the body in white as well as
for the exterior and interior trim. The dramatic shortening of the total development time during the last years needs a much more front loaded development process which has been realized by numerical simulation. The numerical simulation accompanying the design of a car may be divided into three main phases – the concept, the series development and the optimization phase, figure 1. During the concept phase the passive safety concept and its needed packaging space have to be determined. The series development is finished by prototype testing which should confirm the virtual development in an ideal case. Optimization work should close the development before the car’s launch. During these phases the focus of the numerical simulation changes from a more global view to a very detailed local analysis.

Passive safety simulation is fully integrated into the design process within BMW Group. The requirements to meet the function are developed in cooperation with the different suppliers and consulting engineers and then discussed within the design teams, figure 2. Main target is the optimized energy absorption which should help to fulfill legislative and customer demands. Additionally, the virtual supported design helps meeting the requirements of misuse and insurance rating. With the importance of the virtual analysis, the effort has increased during the last ten years. Currently about 13000 simulation are performed on about 850 CPUs per year.

Within the passive safety simulation a lot of different load cases are tested numerically, figure 3. During the last years the number of load cases has been increased significantly. The variety of applications might be divided into three main topics.

Structural Crashworthiness of Whole Car. When frontal, rear and side crash computations are widely used in the development process, compatibility is a growing application. Usually, these models represent most of the body in white, chassis, driveline and exterior trim components. Interior trim components are modeled when the influence on the overall load paths has to be considered, like seats for side crash analysis. These components might be discretized or their masses are smeared over neighbouring parts.

Occupant Protection. The finite element method is mainly used for occupant protection analysis during side impact. The optimization of structural response, interior trim deformation, and airbag behaviour is done in a parallel and iterative process. Because of the high number of needed simulations, a major number of analyses are done in so called substructure runs which uses boundary conditions of the whole car analysis. The same car model as for the side crash computation of the whole car is used. Interior trim and a dummy model are added.

Crashworthiness of Components. A wide range of applications have to be covered by simulation, some of which are mentioned here. These analyses can be performed independently of the whole car crash simulations, but same meshes are used when possible. The discretization of the components is in general much finer than in whole car analysis but the number of represented components is smaller.

The numerical development influences the design of an increasing number of components. When body in white components have been analyzed for long time, the design of exterior trim and interior trim is mainly driven by virtual work today, figure 4. Three main development directions are observed: analysis models are getting much more complex, new or numerically not very well described materials are introduced and, even in full car crash simulations, very local results are needed. For example, the door panel made of reinforced thermoplastic material including the detailed rib and retainer design is today analyzed by using complex side impact load cases.
Finite Element Software Development

During the last years the demands on quality and quantity of numerical simulation have grown extensively. To be able to fulfill these demands a crash vision 2004 has been developed for the analysis software used at BMW Group. This vision covers all the topics related to future numerical crash analysis, and is divided into two main parts – Solver and Process. Within this vision a significant number of different topics are discussed and their requirements defined, figure 5. Based on such discussions, which are done with software suppliers and other partners, the needed software developments are planned and realized.

The solver part of the crash vision is divided into different topics: General Items, Numerical Method, Connecting Techniques Modeling, Material Modeling and Feature Modeling. Within these topics different items and their future requirements are described, figure 6. To meet the future demands on providing numerical results, the turnaround time for a structural analysis should be limited to 12h using 8 to 16 CPUs. To avoid the user’s wrong choice on element types, automatic load case dependent choice of best element type and properties should be included into future software codes.

The process part of the crash vision is divided into the following topics: Load Cases, Preprocessing and Postprocessing, figure 7. Again several items are included into each topic. Automatic load case assembling and validation, as well as automatic load case dependent positioning of barriers and dummy are defined as future requests for crash simulation.

The crash vision 2004 allows for defining the requirements on the used software for the near future. Main topics of software development are material modeling, feature modeling, numerical methods, connecting technique and various topics within new methods, figure 8.

For defining the software requirements and to demonstrate the current status of analysis techniques, an example data base is under development. Basic software and modeling quality is one of the most important demands for doing high-end passive safety simulation in future. Because of this importance, BMW has decided to work closely together with software suppliers to guarantee that the requirements on using the best method available, on highly developed analysis standards, and on an optimized relation between effort and result, are met.

Using ABAQUS/Explicit in Crashworthiness Simulation

The variety of applications and given numbers of topics within the method development show that testing ABAQUS/Explicit for crashworthiness simulations can only be a step by step process. The process is divided into four main parts – the basic testing, a feasibility study, the development of modeling techniques, and the material modeling, figure 9. Today, the focus of this process is on the feasibility study. Some preliminary studies on the basic testing and material modeling have been done.

For testing purposes, test cases are defined and analyses are performed in all cases with the current used crash code and with ABAQUS/Explicit. All results are documented in a web-based documentation. The following examples demonstrate the current status of the testing of ABAQUS/Explicit in crashworthiness simulation.
Basic Testing

Contact. Nearly all of the given examples include severe contact problems to be solved, but special examples are set up to study the contact behaviour in a more detailed way. The aim of the shown example is to study the handling of severe overclosures occurring during the analysis – a situation which happens quite often during crash simulations caused by high contact forces. The usually applied penalty technique for contact handling allows such overclosures. To study this behaviour a shell element is forced by a prescribed displacement to penetrate a surface represented by four other shell elements, figure 10. The contact force is measured. First contact appears depending on the chosen options at a certain contact thickness. The results show that in ABAQUS/Explicit the contact force does increase linearly with the penetration using both the contact pair technique and the newly introduced general contact capability. The current crash code shows different results depending on the chosen contact type. In the current crash code, the detected contact direction changes when the midsurfaces cross each other, and contact is lost when the overclosure gets higher than the contact thickness. In conclusion, ABAQUS/Explicit does not lose contact and contact directions, but the current crash code does. This behaviour may lead to the effect that severe overclosures are prevented from separating again by an opposite contact force and, thus, may destabilize the analysis.

Spotweld. Spotwelds are the most important connecting technique for connecting the components of a body in white. Two test cases for spotwelds are shown in figure 11. The first one is a shear test of two spotwelds connecting three shell element layers. The shear force is introduced by fixing the lowest layer and introducing tension forces on the mid and upper layer in opposite directions. During this test the location of the spotweld has been changed to study mesh influences. The ABAQUS spotweld capability has its advantages in an unconditionally stable formulation and a mostly mesh independent behaviour by using a distributed coupling mechanism for the involved surfaces. Compared to the penalty formulation of other codes, the ABAQUS spotweld does not allow for unintended elongation. The number of involved nodes is dependent on the spotweld diameter and the mesh size. But, a minimal number of nodes for each surface is given. The crushing of a rail geometry demonstrates the stability of the spotweld modeling under severe deformation.

Material Modeling

Thermoplastics. Non-reinforced thermoplastic materials are used for door panels and pillar finishers. Accurate numerical material models are needed to analyze the deformation behaviour of such components for passenger restraint simulation during side crash events. Within a PhD work, a local measurement technique has been developed and a new material model has been formulated for thermoplastic material, Junginger et.al. (2002). Already, the new experimental results used within a classical von Mises material model allow for much better results than have been gained by using the data of a global measurement technique. The measured force deflection curve of a one dimensional tension test is compared in figure 12. The results of the global measurement lead to too early failure prediction. Using the data of the local measurement both in a von Mises model and in the newly introduced model shows good correlation between the experimental and numerical results.
Feasibility Study – Component Crashworthiness

Bumper. A bumper model crashing against a rigid offset barrier is a good example to study contact and element behaviour under severe deformations. The given example, figure 13, includes shell elements for the structural parts and the lining, and solid elements for the bumper foam. Different materials, such as aluminum, thermoplastic, and polymeric foam are included in this model. The bumper is attached to a rigid rectangular block and to a mass point at the center of gravity of the car. The overlapping part of the bumper and the left deformation element is totally crushed. The analysis with ABAQUS/Explicit demonstrates quite well the capability of the new general contact capability and the stability of element and material formulations. A newly introduced distortion control feature for solid elements will additionally stabilize such analyses.

Seat. This example is a standard test case for testing the stability of newly discretized finite element seat models at BMW, figure 14. The seat, which is discretized by shell, rigid, spring, and beam elements, and includes additional connector elements for modeling hinges and multi point constraints for bolts, is attached to the body in white cross rail bars. A rigid surface representing the center console is fixed. Another rigid surface representing the B-pillar is moved against the seat, thus crushing the seat between the two rigid surfaces. The upper part of the seat is deformed severely. The analysis is stable and contact conditions are handled correctly. By default, the general contact capability does include beam elements.

Tires. Correct tire modeling is quite important for a frontal crash analysis, as the tire behaviour influences barrier deformation and passenger compartment intrusion. The current tire models do include a fairly detailed modeling with solid elements discretizing the tire tread, reinforced shell elements and a pressurized interior air volume – usually modeled by using an airbag model with reduced functionality. The current ABAQUS model is set up equivalently except for the pressurized air volume. Here, hydrostatic fluid elements are used, which show a slightly different behaviour as compared to the airbag model. In this example, the tire is fixed at its center and compressed by a rigid surface, figure 15. The results of ABAQUS/Explicit are compared to the current crash code. The overall deformation behaviour is quite similar, contact forces show good correlation as well. The ABAQUS results show higher oscillations because the current crash code provides a prefiltering technique on the contact force output variable. The differences in volume change are negligible.

Door. The example of a pole crashing against a door allows for testing element and contact behaviour, as well as studying the behaviour of the modeling of hinges and locks, figure 16. The door, mainly discretized by shell elements, is attached to the body in white by the hinges and the lock. The part of the hinges and the lock which belong to the body in white are fixed. During the analysis the door undergoes severe deformation normally leading to a failure of the modeled side window. Here, no failure model has been used so that the analysis shows a different deformation behaviour. The behaviour of the hinges is equivalent to the current used crash code. The introduced connector elements allow for a modeling of hinges in a much simpler way and the behaviour is unconditionally stable. The pole test of a door demonstrates again the stability of ABAQUS/Explicit for crashworthiness applications.

Body in White. When the above shown component analyses already demonstrate the general feasibility, a body in white component model does include a substantially higher number of different parts, elements and spotwelds. The first testing example is a body in white crashing against a rigid wall which overlaps the structure totally. In addition to the body in white, some
parts of the bumper and the chassis have been included to this model. The model mainly consists of shell elements and a few thousand spotwelds. The body in white crashes with an initial velocity into the rigid barrier. The deformation behaviour shown in figure 17 correlates well with currently used crash code.

Feasibility Study – Whole Car Crashworthiness

Frontal Impact. As component test cases demonstrate that needed features are provided by ABAQUS/Explicit, a whole car crash analysis for simulating the structural crashworthiness is the next step for test purposes. Here, the USNCAP frontal crash simulation run with a rigid barrier has been done, figure 18 and 19. The model consists of most components of body in white, chassis, powertrain and the frontal bumper. The structural part of the seat has also been included. The number of elements is about 500,000 and the total crash time is about 120 msec. Rebound occurs at about 80 msec. Within this analysis, the contact handling, as well as the modeling of spotwelds and connectors, have been very stable, and the deformation behaviour appears correct. The analysis proves the general feasibility of ABAQUS/Explicit for structural crash analysis.

Productive Application

Head Impact. Head impact according to extended Federal Motor Vehicle Safety Standard No. 201 is a legislative component testing which has to be passed by every automobile sold in the U.S. since September 2002, NHTSA (1995). Legislative demands and the process of simulation have been described in more detail in Gholami et.al. (2002) and Paßmann et. al. (2002). ABAQUS/Explicit has been introduced into the productive application in summer 2002. The main advantages of ABAQUS are the high quality of material modeling, such as the modeling of polymeric foams and of the headform’s vinyl skin, as well as the accuracy of the general contact capability, the stability of the analysis, and the spotweld modeling. The given example, figure 20, demonstrates head impact on target point RP1 in rear pillar target area.

Conclusion

Today, the development of new automobiles relies strongly on finite element simulation with regard to the crashworthiness behaviour. Based on a crash vision for 2004, demands on future software for crash simulation have been defined, and an example data base has been built up to validate software codes. ABAQUS/Explicit has been successfully tested and introduced into the crashworthiness simulation utilizing important new functionality, such as a general contact capability and spotweld modeling feature. In addition, robust libraries of existing features (material models, element formulations, connectors, penalty contact algorithm, time incrementation) and a strong code architecture have been instrumental in enabling this work to be carried out successfully. Today, general feasibility has been proved in many component applications and in whole car crash analysis. Productive usage of ABAQUS/Explicit for head impact analysis has been started.

The development of needed features for occupant protection simulation, an efficient parallel version, and a new barrier model are the next development steps before introducing ABAQUS/Explicit into the productive crashworthiness simulation at BMW Group.
References

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

M. Junginger, H. Werner, R. Paßmann, S. Hiermaier
Characterization and Modeling of Thermoplastic Materials for Crash Applications
CrashMAT 2002, Freiburg, Germany, April 15-16, 2002

NHTSA National Highway Traffic Safety Administration, Department of Transportation
Federal Motor Vehicle Safety Standard No. 201
Occupant Protection, 49 CFR Parts 571, 572 and 589, 1995

R. Paßmann, A. Theobald, W. Jansohn
Simulation des Kopfaufpralls nach FMVSS 201 mit ABAQUS/Explicit
VDI Tagung Berechnung und Simulation im Fahrzeugbau, VDI Berichte, Würzburg 2002
Figures

**Development Process for Passive Safety Function**

- design and packaging freeze
- confirmation of function
- prototype testing
- confirmation of product
- start of series production

**Passive Safety Simulation**

*Figure 1: Passive safety simulation within the different design phases.*
Figure 2: Passive safety simulation within development process.

Figure 3: Load cases within passive safety simulation.
Figure 4: Analyzed components within passive safety simulation.

Figure 5: Crash Vision 2004.
**Passive Safety Simulation**

**Crash Vision 2004 – Solver**

**Solver**
- **General items**: documentation, input structure, user interface, quality assurance.
- **Numerical Method**: turnaround/cost, elements, Newton, time step, mass scaling, contact friction/hard contact, random fields, measuring, pre-and post-processing, stability, substructuring, super-elements, treatment of meshes, temperature, numbering, initial conditions, explicit-implicit coupling, data checking, stochastics, coupling with meshless methods/fluids.
- **Connecting Technique Modeling**: deformable, rigid.
- **Material Modeling**: numerical models, data base.
- **Features Modeling**: dummy, airbag, tires, belts, barriers.

**Structural Computation**
- 12h turnaround on 8 – 16 cpu
- Load case dependent automatic choice of best element type and properties

*Figure 6: Crash Vision 2004 – Solver part.*

**Passive Safety Simulation**

**Crash Vision 2004 – Process**

**Process**
- **Load Cases**
- **Preprocessing**: meaning, folding of airbags, assembling, pooling of summaries, barriers, seat and headform, mixture use of components, stochastics.
- **Postprocessing**: curves, contour fields, cross section, animations.

**Automatic load case assembling and validation**
- Load case dependent automatic positioning of barrier

*Figure 7: Crash Vision 2004 – Process part.*
Figure 8: Software Method Development.

Figure 9: Testing ABAQUS/Explicit for crashworthiness application.
Figure 10: Contact test example for severe overclosures.

Figure 11: Spotweld testing – shear test and behaviour undergoing high deformations.
Figure 12: Material modeling of thermoplastics.

Figure 13: Component crashworthiness testing – bumper testing.
Figure 14: Component crashworthiness testing – seat deformation test.

Figure 15: Component crashworthiness testing – tire behaviour.
Figure 16: Component crashworthiness testing – door pole test.

Figure 17: Component crashworthiness testing – body in white crash test.
Figure 18: Whole car crashworthiness testing - load case USNCAP.

Figure 19: Whole car crashworthiness testing - load case USNCAP.
Figure 20: Component crashworthiness application – head impact according to FMVSS 201.