

Automobile Roof Crush Analysis with Abaqus

Summary

The National Highway Traffic Safety Administration (NHTSA) mandates the use of certain test procedures to determine automobile roof crush resistance. In the test the force-deflection behavior of the roof structure is measured by quasi-statically pressing a precisely positioned rigid plate against the automobile. As part of the design process, the test is often simulated analytically.

As with many quasi-static processes, the roof crush resistance test can be simulated in Abaqus/Standard or Abaqus/Explicit. In this technology brief the modeling techniques used for each analysis product are presented, and it is shown that both products can be used to simulate a roof crush resistance test effectively.

Background

The requirements for roof crush resistance of an automobile are specified in Federal Motor Vehicle Safety Standard (FMVSS) 216. The purpose of the standard is to reduce deaths and injuries resulting from the collapse of the roof into the passenger compartment in a rollover accident.

A schematic diagram of the roof crush resistance test is presented in Figure 1. A force is applied quasi-statically to the side of the forward edge of the vehicle roof structure through a large rigid block. The chassis frame and the car's sills are constrained to a rigid horizontal surface. The force applied to the block and the displacement of the block are recorded throughout the test to characterize the roof crush resistance.

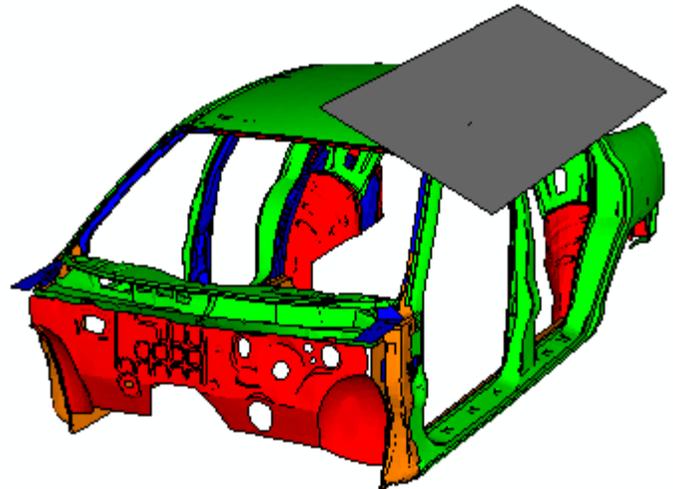
Accurate and efficient finite element modeling of the roof crush resistance test can facilitate the design of safer automobiles as well as reduce development and testing costs.

Finite Element Analysis Approach

Abaqus offers two methods to analyze quasi-static events: an explicit dynamic procedure in Abaqus/Explicit and an implicit static procedure in Abaqus/Standard. The choice of analysis product depends on the application.

Abaqus/Explicit is particularly well suited for the simulation of discontinuous and unstable events. In addition, the general contact capability in Abaqus/Explicit allows for the simplified definition of complex contact conditions.

In Abaqus/Explicit the explicit central difference time inte-



Key Abaqus Features and Benefits

- General "automatic" contact capability in Abaqus/Explicit allows complex contact conditions to be defined easily.
- Automatic stabilization in Abaqus/Standard allows for the simulation of structures experiencing local instabilities in a static procedure.
- Integration of Abaqus/Explicit and Abaqus/Standard allows for the flexibility of reusing the same model for multiple types of analysis procedures.

gration rule is used to advance the solution. The conditional stability of this approach requires the use of small time increments. It can, therefore, be computationally impractical for the modeling of quasi-static events in their natural time scale. Event acceleration techniques must be employed to obtain an economical solution.

Abaqus/Standard is efficient in modeling events with longer durations because the inherent stability of the implicit method allows for the use of relatively large time increments during the solution. The implicit solution procedure differs from the explicit solution procedure in that the solution of the global set of equations requires the convergence of iterations, which can sometimes be challenging.

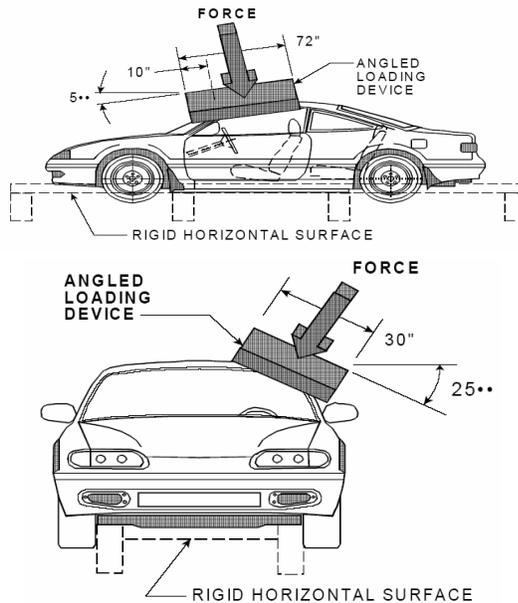


Figure 1: Roof crush resistance test setup (Ref 1).

In this technology brief modeling techniques for each analysis product are presented. It is shown that both products can be used effectively to simulate a roof crush resistance test.

Abaqus allows for the reuse of the same basic model for multiple types of analyses. Models built for one application (Abaqus/Standard or Abaqus/Explicit) can generally be converted for use in the other application with minimal effort. In addition, the flexibility provided by the integration of both analysis products facilitates the import of analysis results from Abaqus/Explicit into Abaqus/Standard and vice versa.

Finite Element Model

The roof crush model is based on the public domain model of a Dodge Neon available from the FHWA/NHTSA National Crash Analysis Center web site (<http://www.ncac.gwu.edu>). The full vehicle model is translated to Abaqus format, and the components necessary for simulating the roof crush test are extracted. The window glass and the components of the interior and exterior trim normally have a negligible effect on the overall roof crush resistance response and are not included. It is also shown that including the front door in the model has a negligible effect. The model geometry is shown in Figure 2.

Connections between different parts of the model are represented using beam MPCs, although mesh-independent spot welds, which offer more general capabilities, could also be used. Shell elements are used to represent all components made of sheet metal. Finite membrane strain shell elements (S4R, S3R) are used to compare the Abaqus/Explicit and Abaqus/Standard results. However, small-strain shell elements (S4RS, S3RS) can and most likely would be used in the Abaqus/Explicit analysis for computational efficiency.

Material properties of all sheet metal components in the model are characterized by Mises plasticity with isotropic hardening. All nodes along the bottom sills and rear wheel housings (on the driver and passenger sides) are constrained to represent a "rigid" floor panel. The loading is displacement driven and is applied at the reference node of a rigid body that models the loading surface. The loading displacement is applied in the direction perpendicular to the rigid surface.

The input data used by Abaqus/Explicit and Abaqus/Standard are very similar; the differences arise in the definition of:

- analysis procedure,
- contact conditions,
- load amplitude, and
- mass scaling.

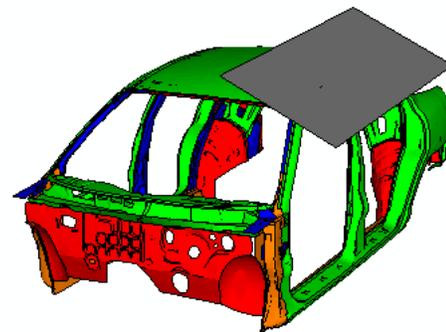


Figure 2: The undeformed shape of the roof crush resistance test model.

Abaqus/Explicit Analysis

As discussed earlier, efficient analysis of quasi-static events using the explicit dynamics procedure requires the use of event acceleration techniques. As the event is accelerated, however, inertial forces may become dominant. The goal is to model the process in the shortest time period in which inertial forces remain insignificant.

Two methods to obtain an economical quasi-static solution with an explicit dynamic procedure are to increase the loading rates and to perform mass scaling. In the first method the duration of the event is reduced artificially by increasing the rate at which the load is applied. In the second method the material density is increased artificially, which leads to an increase of the stable time increment. Both methods are used at the same time for the present analysis.

One approach to determining the extent to which the loading rate can be increased is to study the natural frequencies of the structure using Abaqus/Standard. In a static or quasi-static analysis the lowest eigenmode of a structure usually dominates the response.

Knowing the frequency and the corresponding time period of the lowest mode, you can estimate the time required to obtain a quasi-static response. A starting guideline is to specify a loading time greater than 10 times the period of the lowest eigenmode.

For the roof crush structure with a slight preload by the loading plate, the frequency of the lowest eigenmode is approximately 15.5 Hz, which corresponds to a time period of 65 ms. An analysis time of 400 ms was found to be sufficient to ensure quasi-static loading.

In Figure 3 the force-displacement behavior of the rigid loading plate is plotted. Specifically, the reaction force at the rigid plate reference point (the point controlling the motion of the plate) is plotted against the displacement magnitude of the rigid plate reference point.

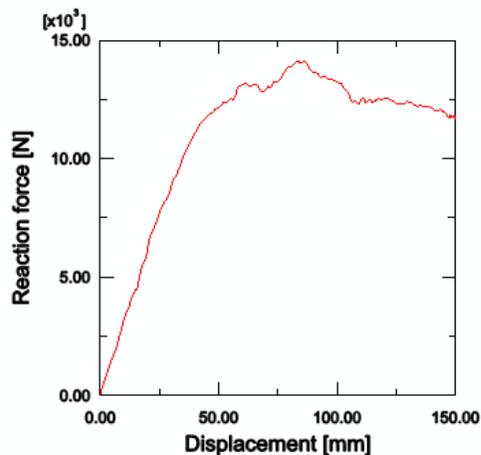


Figure 3: Roof crush resistance curve for Abaqus/Explicit analysis.

As a general rule, to determine whether an analysis is quasi-static, the kinetic energy of the deforming structure should not exceed a small fraction (typically 5%) of its internal energy throughout most of the simulation. In Figure 4 the internal and kinetic energies are plotted.

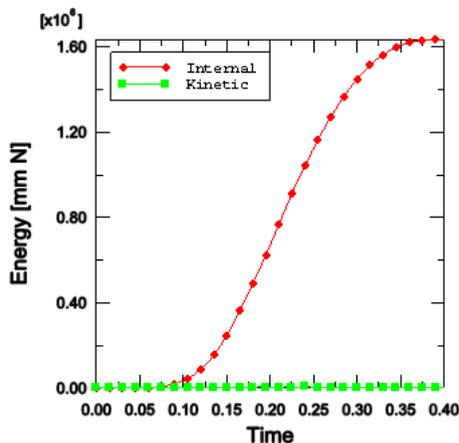


Figure 4: Histories of internal energy and kinetic energy for ABAQUS/Explicit analysis.

Figure 5 shows the final deformed shape of the vehicle structure. In the analysis presented here, general “automatic” contact is defined using an all-inclusive, element-based surface that is defined automatically by Abaqus/Explicit, thus allowing for an easy definition of the contact domain.

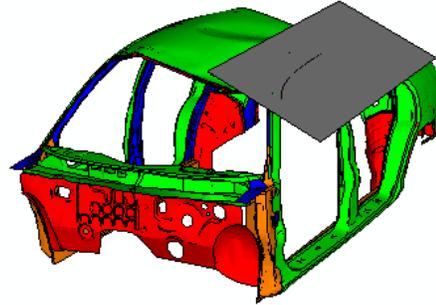


Figure 5: Deformed shape of vehicle structure predicted by Abaqus/Explicit.

To investigate the effect of including additional body components on the overall roof crush resistance, the driver-side front door was added to the Abaqus/Explicit model (Figure 6). The door is assumed to be locked. Figure 7 compares the Abaqus/Explicit analysis results with and without the door and shows that, for the present model, the stiffening of the vehicle structure between the door hinges and the lock has a negligible effect on the overall roof crush resistance.

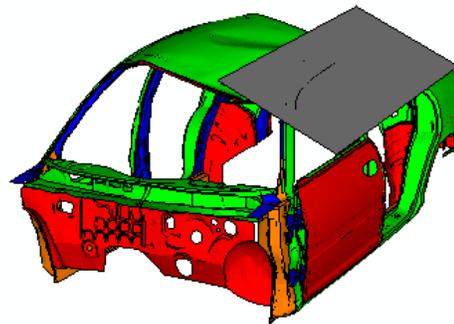


Figure 6: Deformed shape of vehicle structure with door predicted by Abaqus/Explicit.

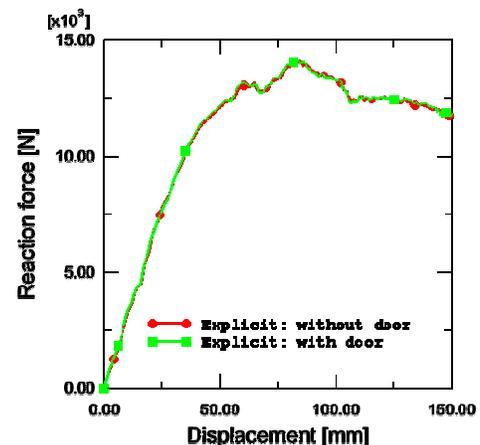


Figure 7: Roof crush resistance response predicted by ABAQUS/Explicit for vehicle structure with and without door.

Abaqus/Standard Analysis

The static procedure in Abaqus/Standard neglects inertial effects and is, thus, a natural choice to model quasi-static events.

As the automobile body is loaded, the roof structure may exhibit numerous local instabilities; such instabilities can cause convergence problems for an implicit solution method. Abaqus/Standard offers a mechanism to stabilize this class of problems by adding volume-proportional viscous damping to the model. This stabilization is used for the present analysis.

Abaqus/Standard offers a robust contact pair algorithm that requires the definition of all potential contact interactions between different surfaces in the model. To minimize the expense of the contact calculations and to simplify the model definition, contact has been defined only between the rigid plate and the regions of the body that the plate is likely to contact. Additional surface-based tie connections have been specified to approximate contact conditions in the regions close to the rigid plate. Although such connections impose constraints between the tied surfaces, the effect on the overall response is minimal since these surfaces are unlikely to separate during the simulation. Figure 8 shows the final deformed configuration of the vehicle structure at the end of the static analysis.

Comparisons and Conclusions

Figure 9 compares the roof force-displacement response predicted by Abaqus/Explicit and Abaqus/Standard.

The force-displacement responses predicted by both products are very similar except in the latter part of the analyses, when contact begins to play a dominant role. Contact conditions were simplified in the Abaqus/Standard model, facilitating a reduction in computing time but introducing some inaccuracy in the final solution. The deformed configurations from both analyses are shown in Figure 10.

The results of the Abaqus/Standard analysis can be improved further by considering additional contact interactions in the vehicle structure. The Abaqus/Explicit model uses more complete contact definitions with contact defined for the entire model rather than for the most critical regions.

The model under consideration is based on a public domain FEA model and does not represent an actual production vehicle. No information was available to verify the material properties, shell thicknesses, spot weld spacing, and other details that must be specified. These properties have a significant influence on the model behavior.

The results presented here demonstrate that both Abaqus/Explicit and Abaqus/Standard can be used effectively to simulate a roof crush resistance test. The choice of analysis product depends on several factors such as

rate of loading, complexity of contact conditions, etc. Nevertheless, a unified model that can be used with both analysis products enables efficient evaluation of both possible solutions. As passenger protection in rollover accidents assumes increasing importance, evaluation of different analysis solutions may be necessary to obtain a thorough understanding of the roof crush resistance of the structure.

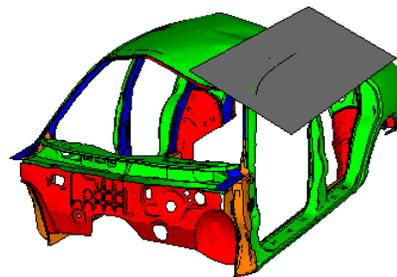


Figure 8: Deformed shape of vehicle structure predicted by Abaqus/Standard.

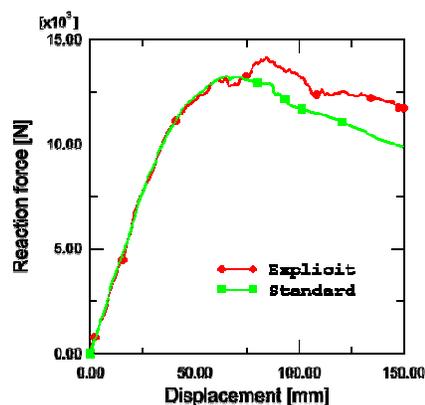


Figure 9: Roof crush resistance response predicted by Abaqus/Standard and Abaqus/Explicit.

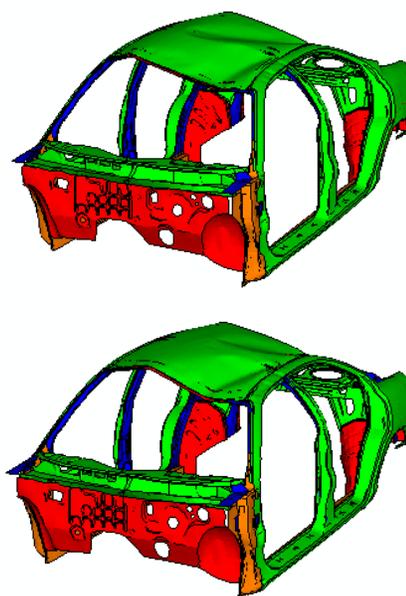


Figure 10: Final deformed configurations predicted by Abaqus/Explicit (top) and Abaqus/Standard (bottom).

Acknowledgements

The model used in this paper is based on the public domain Dodge Neon model available through the FHWA/NHTSA National Crash Analysis Center (NCAC) web site (<http://www.ncac.gwu.edu>).

References

1. Laboratory Test Procedure for FMVSS 216 Roof Crush Resistance—Passenger Cars, TP-216-04, U.S. Department of Transportation National Highway Traffic Safety Administration, August 1989.
2. Fichtinger, G., and R. Paßmann, BMW Group, and F. G. Rammerstorfer, Vienna University of Technology, “Roof Indentation Simulation with Abaqus,” Abaqus Users' Conference, Maastricht, the Netherlands, June 2001, pp. 203–216.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.13 documentation references:

- Getting Started with Abaqus
 - “Quasi-Static analysis with Abaqus/Explicit,” Chapter 13
 - “Comparison of implicit and explicit procedures,” Section 2.4
- Abaqus Analysis User's Guide
 - “Static stress analysis,” Section 6.2.2
 - “Explicit dynamic analysis,” Section 6.3.3
- Abaqus Example Problems Guide
 - “Unstable static problem: reinforced plate under compressive loads,” Section 1.2.5

About SIMULIA

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

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

Copyright © 2007 Dassault Systèmes

