Prediction of B-Pillar Failure in Automobile Bodies

Summary

The B-pillar is an important load carrying component of any automobile body. It is a primary support structure for the roof, and is typically a thin-walled, spot-welded, closed-section structure made from high strength steels.

As part of the validation process, the B-pillar can be experimentally loaded at quasi-static rates until failure[†]. The force and displacement of the impactor are measured to get valuable insight into the stiffness characteristics of the structure.
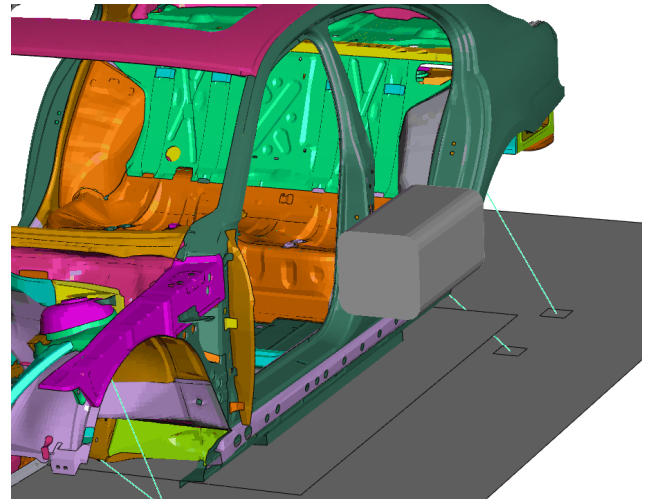
During the past two decades, crashworthiness simulation of automotive structures has proven to be remarkably good, largely because the finite element codes being used can accurately predict the plastic bending and stretching deformation mechanisms that occur in stamped metal parts.

Abaqus/Explicit offers a general capability for predicting the onset and evolution of damage in ductile metals. The Müschenborn-Sonne forming limit diagram (MSFLD) damage initiation criterion allows for the prediction of necking instability in sheet metal. When combined with appropriate damage evolution criteria, sheet metal rupture initiated by necking can be captured in an Abaqus/Explicit simulation. In this Technology Brief, the MSFLD criterion is employed to investigate the failure of a B-pillar structure. Abaqus/Explicit results are shown to match well with experimental data provided by BMW.

Background and Analysis Approach

The B-pillar failure test is not required by any federal regulations. The purpose of this test is to determine the load carrying capacity and stiffness of the structure, which is deliberately loaded to failure. A picture of the B-pillar experimental test shown in Figure 1. In the figure, the B-pillar is the vertical, orange member of the automobile body structure.

The geometry and mesh of the finite element model are shown in Figure 2. All the sheet metal parts are modeled with small-strain shell elements for computational efficiency. Parts are joined together with mesh independent spot welds. Turnbuckles, which constrain the vehicle body to the rigid floor, are modeled with truss elements.

† *The testing has been performed solely for validating crash simulation software and does not represent or suggest any actual internal design criteria.*

Key Abaqus Features and Benefits

- Ductile metal progressive damage and failure modeling with the Müschenborn-Sonne forming limit diagram damage initiation criterion for rate-dependent elastic-plastic material

- Mesh independent spot welds

- Abaqus/Explicit automatic general contact

The experimental loading is applied quasi-statically, at a rate of 2 mm/s. An Abaqus/Explicit quasi-static analysis conducted in its natural time scale would result in an excessive run time. To increase the efficiency of the analysis, the loading event is accelerated by prescribing a ve-
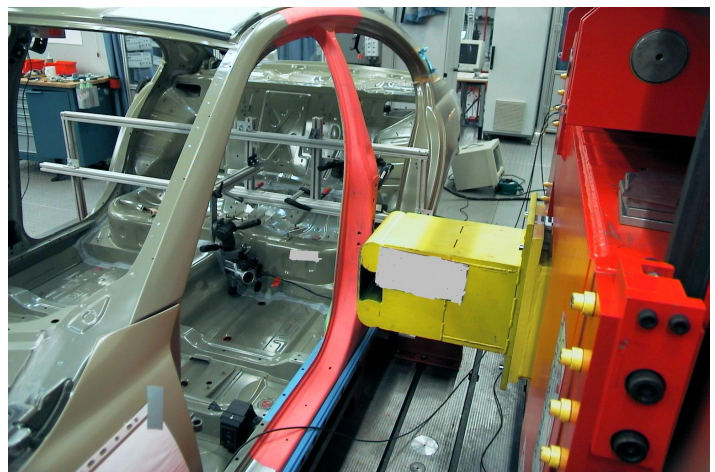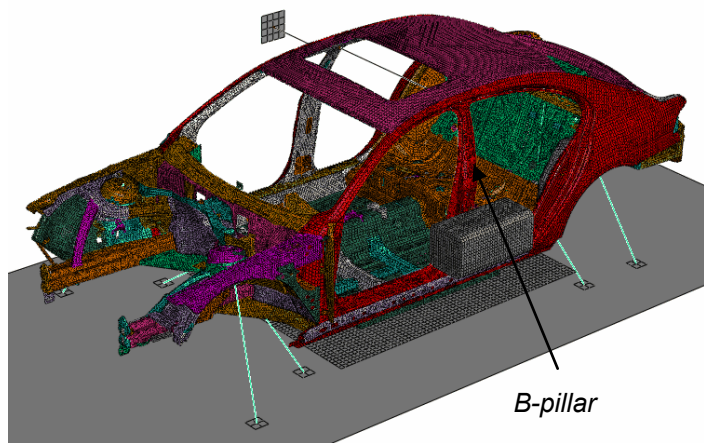

Figure 1: B-pillar experimental test (photo courtesy BMW Group)

Figure 2: Model geometry and mesh

Figure 3: Force-displacement history – experimental result and Abaqus/Explicit without MSFLD initiation criterion

locity boundary condition of 2.5 m/s to the reference node of the rigid impactor.

A tensile load of 3 kN is applied to each turnbuckle in the first 10 ms to pre-stress the structure. To bring the structure into equilibrium, no load is applied on the B-pillar for the next 10 ms. The velocity boundary condition is then applied to the impactor. The impactor reaction force and displacement histories are monitored. General contact is defined for the entire model.

The material properties for all the sheet metal components are characterized by Mises plasticity with isotropic hardening. Three separate damage initiation criteria are used; ductile, shear, and MSFLD. Each damage initiation criteria has an associated displacement-based damage evolution criteria. The results that follow will highlight the importance of including the MSFLD criteria in the simulation of sheet metal rupture associated with necking.

Results and Discussion

An initial simulation was done with only the shear and ductile damage initiation criteria; the MSFLD criterion was not included. A mesh of typical size used in crashworthiness analyses (characteristic element length of 6-8 mm) was employed.

Figures 3 and 4 show the resulting force-displacement curve and the deformed shape respectively. The force-displacement curve from the simulation compares reasonably well with that from the experiment. However, in the absence of the MSFLD initiation criterion, the simulated structure does not fail completely and continues to carry load throughout the analysis.

In a second simulation using the same mesh, the MSFLD initiation criterion was included in the material model, so that all three initiation and evolution criteria were active. Figures 5 and 6 show the force-displacement curve and the deformed shape, respectively, of the experiment and the simulation. The improvement in the simulation results is clear. Representative Mises stress and equivalent plastic strain results are presented in Figure 7, showing how
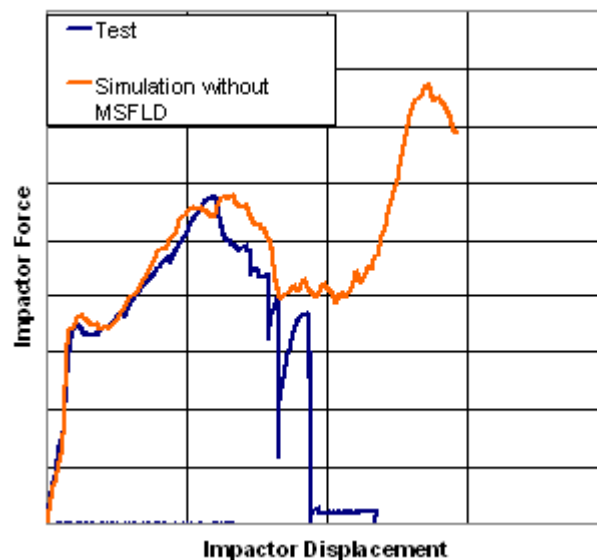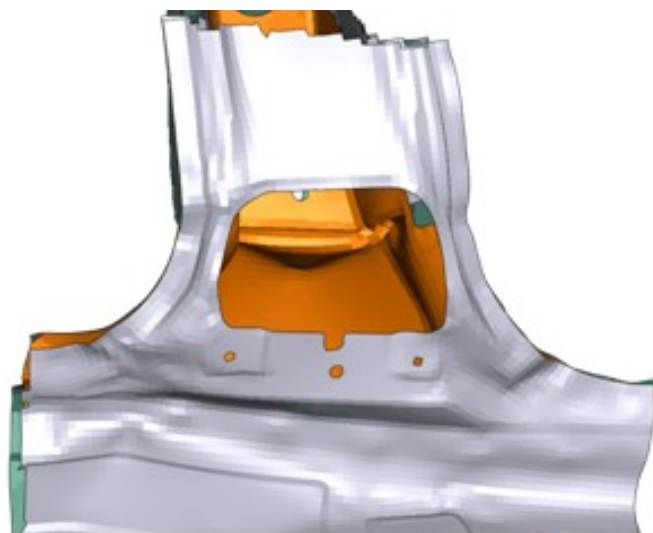

Figures 4a, b: Experimental result and Abaqus/Explicit simulation without MSFLD initiation criterion. In this view, the impactor is behind the B-pillar (photo courtesy BMW Group).
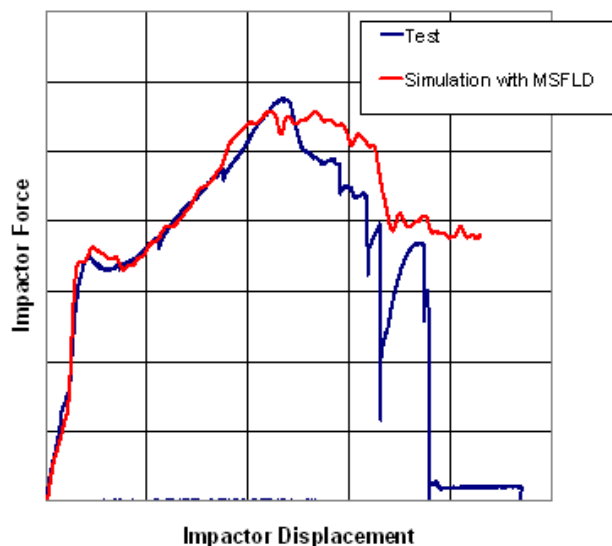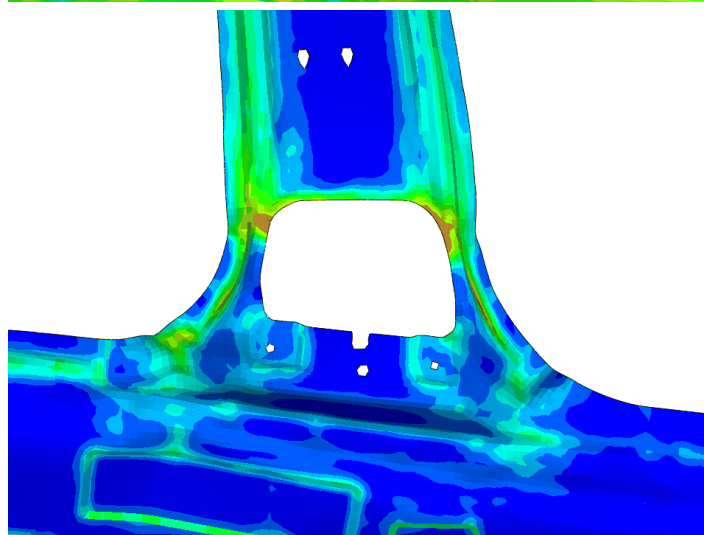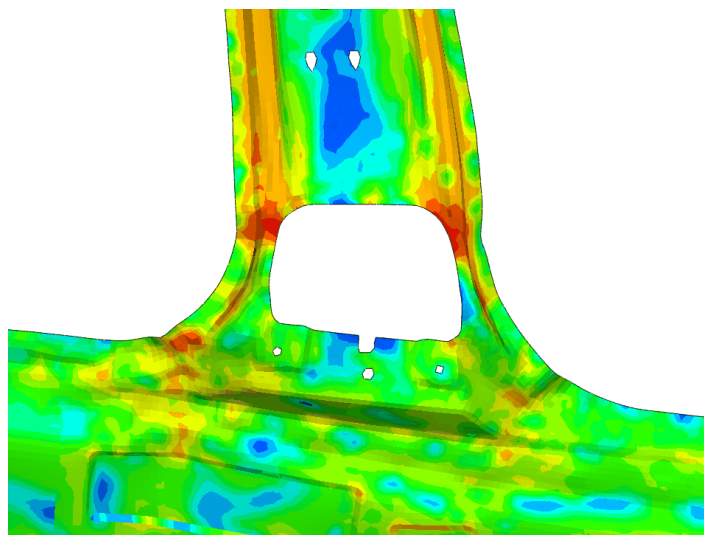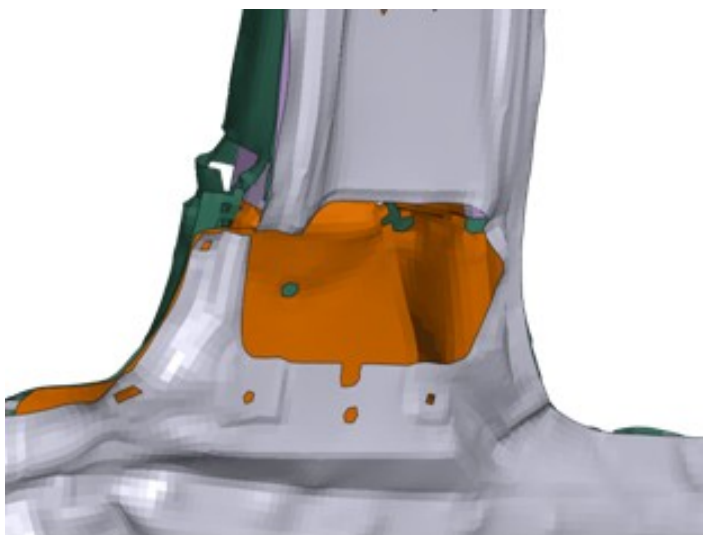
Figure 5: Force-displacement history – experimental result and Abaqus/Explicit with MSFLD initiation criterion

Figures 6a, b: Experimental result and Abaqus/Explicit simulation including MSFLD initiation criterion. In this view, the impactor is behind the B-pillar (photo courtesy BMW Group).

Figures 7a, b: Representative Mises stress and equivalent plastic strain results, Abaqus/Explicit simulation including MSFLD initiation criterion

the development of the crack initiation zone is captured by the simulation.

For extremely large displacement of the impactor, a mesh with regular element sizing does not capture the complete separation of the structure that is observed in the experiment. To evaluate the effect of mesh refinement on the crack propagation and failure, the region of the B-pillar where the crack initiates was re-meshed, with an average local element size of 1.5 mm. The locally refined mesh is shown in Figure 8.

Figure 9 shows the comparison of the force-displacement curves for the two analyses. Figure 10 shows the separation of the B-pillar in the simulation with the refined mesh and in the experiment. The results with the refined mesh show improved correlation with the physical test. The results indicate that a mesh size of 6 to 8 mm is appropriate to predict the area of stress concentration and the crack initiation, but a refined mesh may be necessary to predict crack propagation accurately.
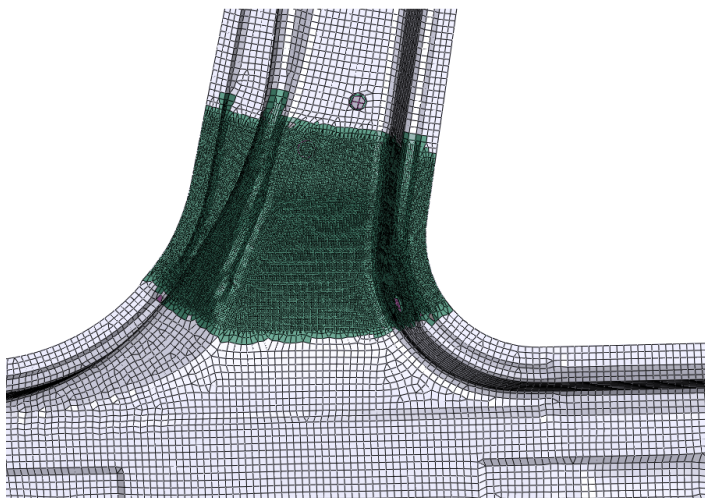
Figure 8: Locally refined B-pillar mesh, with average refined element size of 1-2 mm
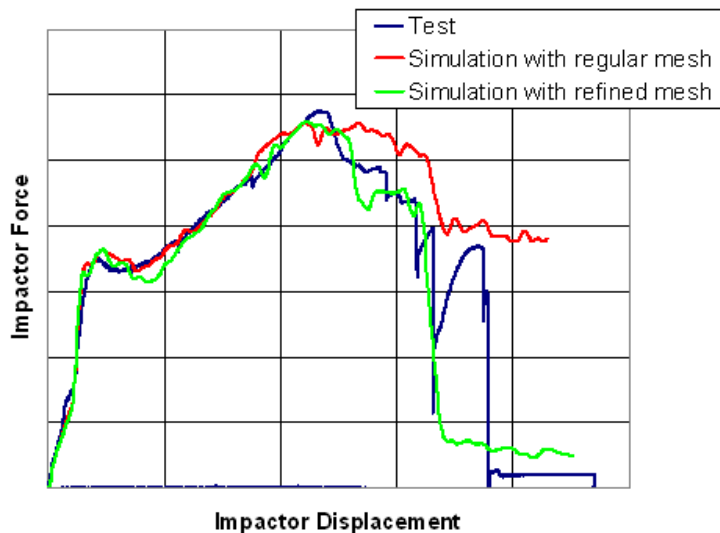

Figure 9: Force-displacement history – experimental result and Abaqus/Explicit with MSFLD initiation criterion and refined mesh
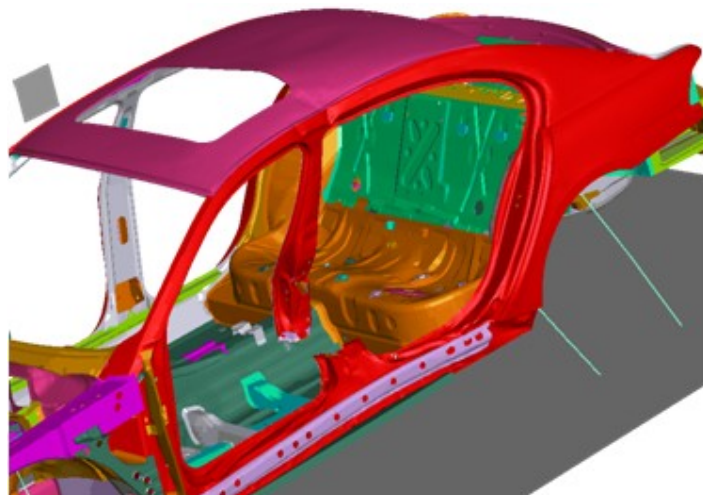

Figure 10: B-pillar final separation: experiment (left) and Abaqus/Explicit with MSFLD initiation criterion and refined mesh (photo courtesy BMW Group)

Conclusion

It has been shown that Abaqus/Explicit can be used to simulate the extreme deformation, failure, and contact conditions that arise during the experimental determination of B-pillar quasi-static strength. The damage initiation and evolution capabilities of Abaqus/Explicit permit the accurate determination of failure in sheet metal associated with necking instability.

Acknowledgements

Dassault Systèmes SIMULIA Corporation would like to thank BMW Group for providing the experimental results and the finite element model used in these simulations.

References

1. A. Pickett, Th. Pyttel, F. Payen, F. Lauro, N. Petrinic, H. Werner, J. Christlein, "Failure prediction for advanced crashworthiness of transportation vehicles", Int. Journal of Impact Engineering, Vol. 30, Issue 7, 2004, pp. 853-872.

2. H. Werner, H. Dell, G. Metzmacher, L. Kessler, A. Heath, "Methodology, Validation and Application of a Failure Model Based on Transient Forming Limit Curves for Coupled Stamping and Crash Processes as Part of the IM-PACT Project", 13th European Conference and Exhibition on Digital Simulation for Virtual Prototyping, Virtual Manufacturing and Virtual Environment. Mainz, Germany, 16.-17.Oct.2003.

3. Müschenborn, W., and H. Sonne, "Influence of the Strain Path on the Forming Limits of Sheet Metal," Archiv fur das Eisenhüttenwesen, vol. 46, no.9, pp. 597–602, 1975.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.11 documentation references:

- Analysis User's Manual
    - "Damage initiation for ductile metals," Section 23.2.2
    - "Damage evolution and element removal for ductile metals," Section 23.2.3
    - "Defining general contact interactions in Abaqus/Explicit," Section 34.4.1
    - "Mesh-independent fasteners," Section 33.3.4

3DS SIMULIA