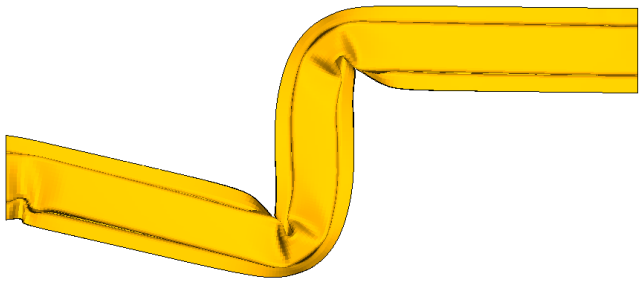
Quasi-Static Collapse of Spot-Welded, Thin-Walled Curved Beams

Summary

Spot-welded, thin-walled curved beams, which constitute the main structural members in many automobile and other ground vehicle body structures, play a significant role in absorbing energy during a collision. Due to their extensive use, it is important to study the collapse characteristics of these curved members (Ref. 1). Abaqus/Explicit can be used effectively to simulate the quasi-static collapse of spot-welded structural members accurately.



Key Abaqus Features and Benefits

- The ability to define contact between many bodies easily using the general "automatic" contact algorithm in Abaqus/Explicit. The general contact capability allows very general surface topologies to be included in the contact domain with very few restrictions on the types of surfaces involved.
- Mesh-independent spot welds (fasteners). Spot welds can be located anywhere between two parts that are to be connected regardless of the mesh. Mesh-independent spot welds couple, in an average sense, the motion of all surface nodes lying within a region of influence to the motion of the spot weld.

Background

Spot-welded, thin-walled structural members contribute significantly to the ability of an automobile body to absorb energy during impact. The front-side and rear-side structural members generally have a curved profile to avoid interference with other vehicle components such as the engine, fuel tank, etc. These thin-walled curved members must be designed appropriately to ensure the crashworthiness of a vehicle. It is essential to study the collapse and postbuckling response of these structural members to arrive at an optimal design from a structural and safety point of view.

Quasi-static collapse and postbuckling behavior are unstable phenomena that can be very difficult to analyze using an implicit finite element method. Further complicating the analysis are the complex contact conditions involved. In addition to experiencing self-contact, the hat and plate portions of the beam will contact one another. Abaqus/Explicit is ideally suited for this class of problems.

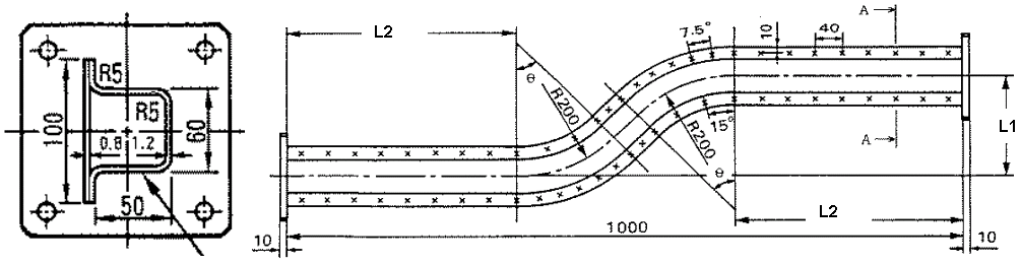
Honda R&D has extensively studied the collapse characteristics of thin-walled curved beams with a closed-hat section in a number of carefully performed experiments (Ref. 1). Figure 1 shows the profile and major dimensions of the test specimens. Test pieces were prepared by spot welding the flat plate section to the hat-shaped section. Mild steel (SPC) sheets of 0.8 mm and 1.2 mm were used for the flat- and hat-shaped sections, respectively.

Figure 2 shows a schematic diagram of the equipment and test specimens used in the collapse test. Two test specimens are arranged in such a way that the axes of the beams maintain lateral symmetry. The test specimens are loaded quasi-statically, and the load-displacement behavior is determined from load cell and potentiometer measurements.

Finite Element Analysis Approach

The geometry for the beam models was created using Abaqus/CAE. Figure 3 shows the finite element meshes used in the quasi-static simulation and a cross-sectional view of the beam. Reduced-integration shell elements employing stiffness hourglass control have been used to discretize the models. The constitutive behavior of the mild steel (SPC) is represented by a rate-dependent Mises plasticity model.

The spot weld locations on the beams are represented with red dots in Figure 3. The spot welds have been modeled using mesh-independent fasteners, which significantly simplifies the modeling of parts to be joined. With this capability, spot welds can be located anywhere between the two parts that are to be connected, regardless of the mesh configuration.



Specimen	Curve angle, q	L1 (mm)	L2 (mm)	Total length (mm)
15° frame	15°	194	150	1000
30° frame	30°	316	150	1000
45° frame	45°	346	150	1000

Figure 1: Profile and major dimensions of test specimen. Dots represent spot welds (Ref. 2).

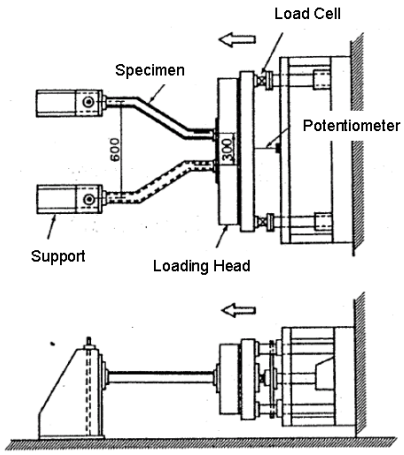


Figure 2: Schematic diagram of test setup (Ref. 2).

The attachment to each of the parts being connected is distributed to several nodes in the neighborhood of the attachment points. Mesh independence is achieved by the automatic generation of distributed coupling constraints in the vicinity of the attachment points. Rather than requiring that nodes from adjoining parts be located

at the weld point, the connection is distributed via distributed coupling constraints to several nodes in the neighborhood of the attachment points. Each distributed coupling constraint smears the connection formed by the spot weld by distributing the transmission of forces and does not impose kinematic constraints between the nodes in the neighborhood of the attachment points.

As shown in Figure 2, the left end of the model is completely constrained and the right end is loaded at a rate that will ensure quasi-static collapse. The general contact algorithm in Abaqus/Explicit is used for this analysis. The entire model has been included in the general contact domain. Surfaces specified in the general contact domain can span unattached bodies and can include rigid and deformable regions with mixed element types. The general contact algorithm enforces contact constraints using a penalty contact method that works very well with mesh-independent fasteners, connectors, and other type of constraints.

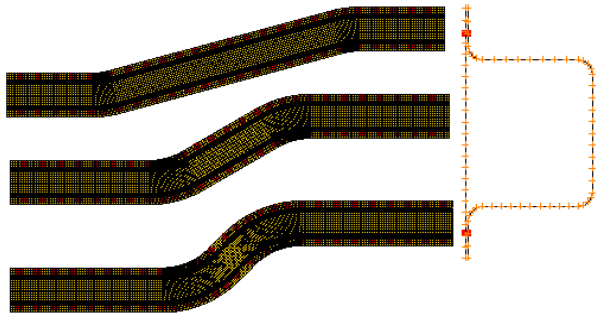


Figure 3: Undeformed model and cross-section for 15°, 30°, and 45° frames. Red dots represent spot weld locations.

Results and Conclusions

The analysis results show that the ratio of total kinetic energy of the model to internal energy is less than 3% throughout the simulation, which indicates that a quasi-static solution was achieved. The deformed shape of the 15° frame at a stroke of 200 mm is shown in Figure 4.

Deformed configurations for the three geometries obtained from the Abaqus/Explicit dynamic simulation are shown in Figure 5. Abaqus predicts the deformation mode accurately in all cases. The onset of collapse

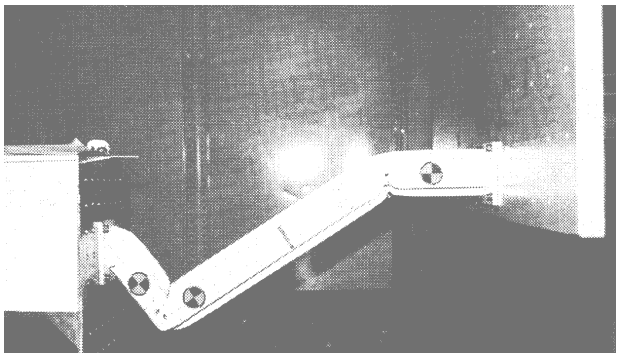


Figure 4: 15° frame at 200 mm stroke (Courtesy: Honda R&D).

(plastic hinge formation) and postbuckling response predicted by Abaqus match well with experimental observation.

Comparisons of the load-displacement behavior of the finite element model and the experimental test specimen are shown in Figures 6, 7, and 8.

Table 1 compares the experimental peak load and mean load values to those obtained from Abaqus. Mean load has been computed based on the total energy absorbed at a stroke of 200 mm. These results and Figures 6, 7, and 8 clearly show that the Abaqus/Explicit predictions exhibit good correlation with experimental results.

Analysis results can be transferred between Abaqus/Standard and Abaqus/Explicit. With this capability, the quasi-static collapse analysis simulation outlined above can be included in a multi-stage analysis that includes the beam manufacturing effects.

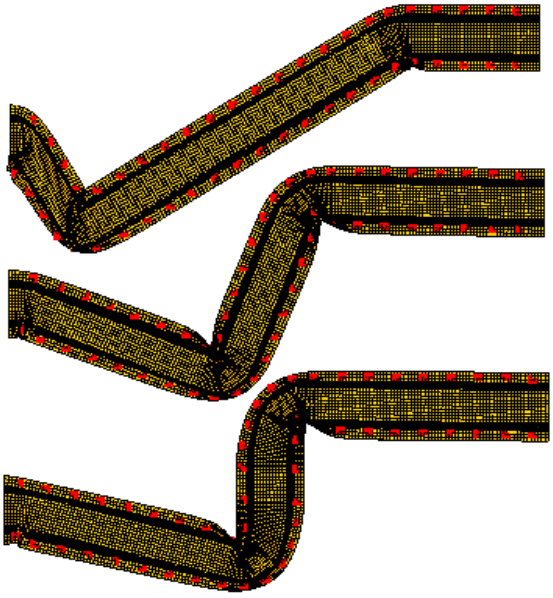


Figure 5: Deformed configurations of 15°, 30°, and 45° frames at 200 mm stroke.

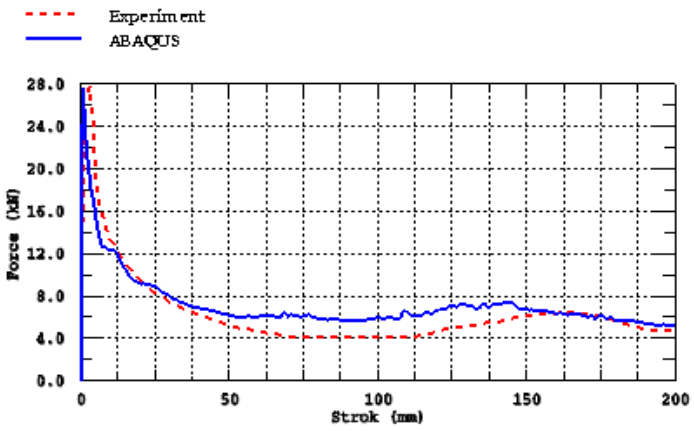


Figure 6: Load-displacement behavior of 15° frame.

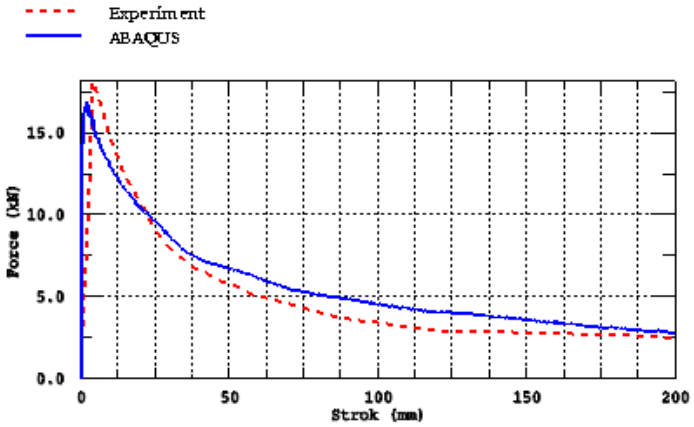


Figure 7: Load-displacement behavior of 30° frame.

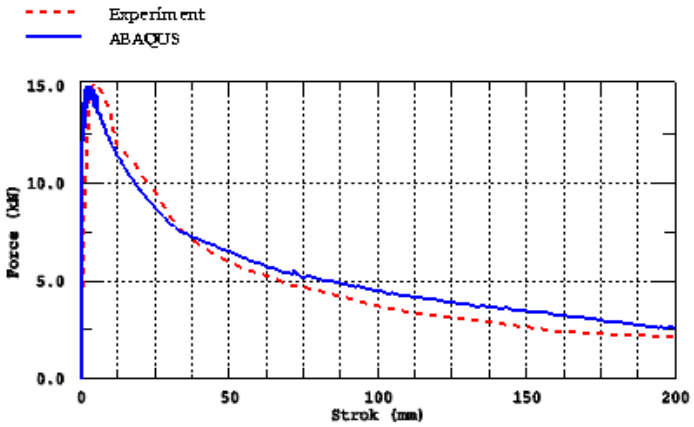


Figure 8: Load-displacement behavior of 45° frame.

Table 1: Comparison of peak load and mean load obtained from Abaqus and experiment.

Specimen	Peak load (KN) Abaqus	Peak load (KN) Experiment	Mean load (KN) Abaqus	Mean load (KN) Experiment
15° frame	27.5	27.7	6.78	6.32
30° frame	17.0	18.1	5.63	4.93
45° frame	15.1	15.2	5.46	4.96

Acknowledgements

Dassault Systèmes SIMULIA Corp. would like to thank Mr. Kenji Takada, Honda R&D, Japan, for providing experimental results.

References

1. Tomizawa, H., Y. Ohkami, K. Motomura, M. Shimamura, M. Usuda, and K. Takada, "Collapse of Thin-Walled Curved Beam with Closed-Hat Section – Part 1: Study on Collapse Characteristics", SAE Technical Paper Series, 900460, International Congress and Exposition, Detroit, Michigan, February 26–March 2, 1990.
2. Society of Automotive Engineers of Japan, Inc., *Structural Strength Commission: Collapse Characteristics of Thin-walled Curved Beams*, vol. 1, Test Methods and Results, pp. 1–222, 1986.

Abaqus References

For additional information on the capabilities and techniques outlined above, see the following references to the Abaqus 6.11 documentation:

- Analysis User's Manual
  - "Explicit dynamic analysis," Section 6.3.3
  - "Mesh-independent fasteners," Section 33.3.4
  - "Defining contact interactions," Section 34
- Example Problems Manual
  - "Buckling of a column with spot welds," Section 1.2.3

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 Suresnes, 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