Buckling of a column with spot welds

This example illustrates both a static and dynamic collapse of a steel column constructed by spot welding two channel sections.

This example is intended to illustrate the modeling of spot welds using mesh-independent spot weld modeling capabilities in Abaqus.

This page discusses:

Products: Abaqus/Standard Abaqus/Explicit Abaqus/CAE

Problem description

The pillar is composed of two columns of different cross-sections, one box-shaped and the other W-shaped, welded together with spot welds (Figure 1). The top end of the pillar is connected to a rigid body, which makes the deformation of the pillar easy to control by manipulating the rigid body reference node. The box-shaped column is welded to the W-shaped column with five spot welds on either side of the box-shaped column.

The columns are both composed of aluminum-killed steel, which is assumed to satisfy the Ramberg-Osgood relation between true stress and logarithmic strain,

$$\varepsilon = \sigma/E + (\sigma/K)^n,$$

where Young's modulus (E) is 206.8 GPa, the reference stress value (K) is 0.510 GPa, and the work-hardening exponent (n) is 4.76. In the present Abaqus analyses the Ramberg-Osgood relation is approximated using elastic and plastic material properties. The material is assumed to be linear elastic up to a yield stress of 170.0 MPa, and the stress-strain curve beyond the yield stress is defined in piecewise linear segments using plastic material properties. Poisson's ratio is 0.3.

The spot welds are modeled in both Abaqus/Standard and Abaqus/Explicit using the mesh-independent fastener capability. Connector elements with CARTESIAN and CARDAN sections are used to define deformable fasteners. Alternatively, a BUSHING connection type could have been used. The element set that contains the connector elements is referenced in the mesh-independent fastener. The spot welds at nodes 5203, 15203, 25203, 35203, and 45203 are all located on the positive *z*-side of the box-shaped column, with node 5203 at the bottom end of the column and node 45203 at the top end of the column (see Figure 2). Spot welds at nodes 5211, 15211, 25211, 35211, and 45211 are all located on the negative *z*-side of the box-shaped column, with node 5211 at the bottom end of the column and node 45211 at the top end of the column. The surfaces of the box-shaped column and the W-shaped column are specified in the mesh-independent fastener. The spot welds are defined with a diameter of .002 m. The deformable behavior in the fastener is modeled using connector elasticity, with an elastic spring stiffness of 2×10^{11} N/m in translational as well as rotational components. For the Abaqus/Explicit analysis, spot weld damage and failure are modeled using connector damage behavior. A force-based coupled damage initiation criterion that uses a connector potential with both connector force and connector moment ingredients is used. (For further description of the connector potential used, see the spot weld example in Connector Functions for Coupled Behavior.) Damage initiates when the value of the potential exceeds 2×10^{5} N. A post-damage-initiation equivalent displacement of 1×10^{-7} m is allowed. Once the post-damage-initiation equivalent displacement in a spot weld reaches this value, the spot weld ceases to carry any load. Both the continuum and structural coupling capabilities are used to define the fasteners.

A Python script is included that reproduces the model using the Scripting Interface in Abaqus/CAE. The script creates and assembles Abaqus/CAE parts and uses discrete fasteners to model the spot welds. The script creates an Abaqus/Standard model that is ready to be submitted for analysis from the Job module. The discrete fasteners created by the script result in the following differences compared with the mesh-independent, or point-based, fasteners used by the example input files:

- When you submit the Abaqus/CAE job for analysis, the discrete fasteners created by the Python script generate coupling constraints and distributing coupling constraints in the input file, together with connector elements. The example input files use mesh-independent fasteners to model point-based fasteners using connector elements.
- You must define the radius of influence when you create a discrete fastener using Abaqus/CAE. In contrast, the example input files allow Abaqus to compute a default value of the radius of influence based on the geometric properties of the fastener, the characteristic length of connected facets, and the type of weighting function selected.
- The input files share nodes between the pillar and the rigid body. To achieve similar behavior, the Python script creates tie constraints between the pillar and the rigid body.

For a description of the differences between discrete fasteners and point-based fasteners in Abaqus/CAE, see About fasteners.

Loading

The bottom of the pillar is fully fixed. In the Abaqus/Standard analysis the reference node for the rigid body at the top of the pillar moves 0.25 m in the *y*-direction, thus loading it in compression, together with a displacement of .02 m in the *z*-direction that shears it slightly. At the same time, the end of the pillar is rotated about the negative *z*-axis by 0.785 rad and rotated about the negative *x*-axis by 0.07 rad.

In the Abaqus/Explicit analyses the reference node for the rigid body at the top of the pillar moves at a constant velocity of 25 m/sec in the *y*-direction, thus loading it in compression, together with a velocity of 2 m/sec in the *z*-direction that shears it slightly. At the same time, the end of the pillar is rotated about the negative *z*-axis at 78.5 rad/sec and rotated about the negative *x*-axis at 7 rad/sec. This loading is applied by prescribing the velocities of the rigid body reference node that is attached to the top end of the compound pillar.

The analysis is carried out over 10 milliseconds.

Results and discussion

Figure 3 shows the deformed shape of the pillar after 5.0 msec in the Abaqus/Explicit analysis. Figure 4 shows the deformed shape of the pillar after 10.0 msec. The spot welds in the mesh-independent Abaqus/Explicit analysis undergo damage and fail. For the current choice of parameters for the connector damage model, it is found that damage initiates in the spot welds at nodes 15203 through 45203 on the positive side of the box-shaped column and at nodes 15211 through 45211 on the negative side of the box-shaped column. However, the post-damage-initiation displacement is sufficient to cause ultimate failure of the spot welds at nodes 15203, 25203, 15211, and 25211 only. Figure 5 illustrates the undamaged connector force CTF3 in the spot welds associated with reference nodes 25203 and 25211 as computed in the Abaqus/Standard analyses. Figure 6 illustrates the damaged connector force CTF3 in the spot welds associated with reference nodes 25203 and 25211 as computed in the Abaqus/Explicit analyses. Forces in both spot welds drop to zero when ultimate failure occurs in the Abaqus/Explicit analyses. Figure 7 shows the time history of the total kinetic energy, the total work done on the model, the total energy dissipated by friction, the total internal energy, and the total energy balance.

Input files

pillar_fastener_xpl.inp

Input data for the Abaqus/Explicit mesh-independent spot weld analysis.

pillar_fastener_structcoup_xpl.inp

Input data for the Abaqus/Explicit mesh-independent spot weld analysis using structural coupling in the fastener definitions.

pillar_fastener_std.inp

Input data for the Abaqus/Standard mesh-independent spot weld analysis.

pillar_fastener_structcoup_std.inp

Input data for the Abaqus/Standard mesh-independent spot weld analysis using structural coupling in the fastener definitions.

pillar_fastener_smslide_std.inp

Input data for the Abaqus/Standard mesh-independent spot weld analysis using small-sliding contact with shell thickness taken into account.

pillar_rest.inp

Input data used to test the restart capability with spot welds.

Python scripts

pillar_fastener_std.py

Script that creates a model with discrete fasteners using Abaqus/CAE.

Figures

Figure 1. Initial configuration of the compound pillar.

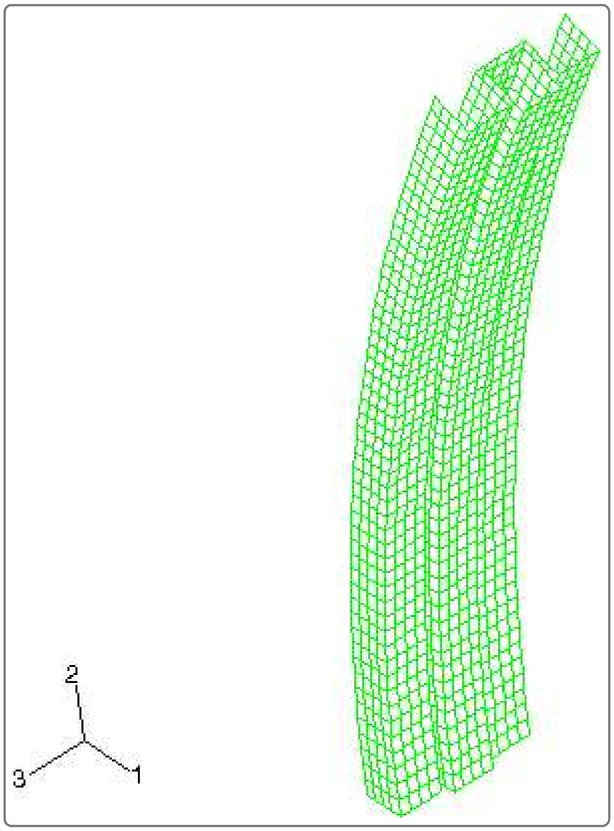

Figure 2. Initial configuration of the box-shaped column showing spot welds.

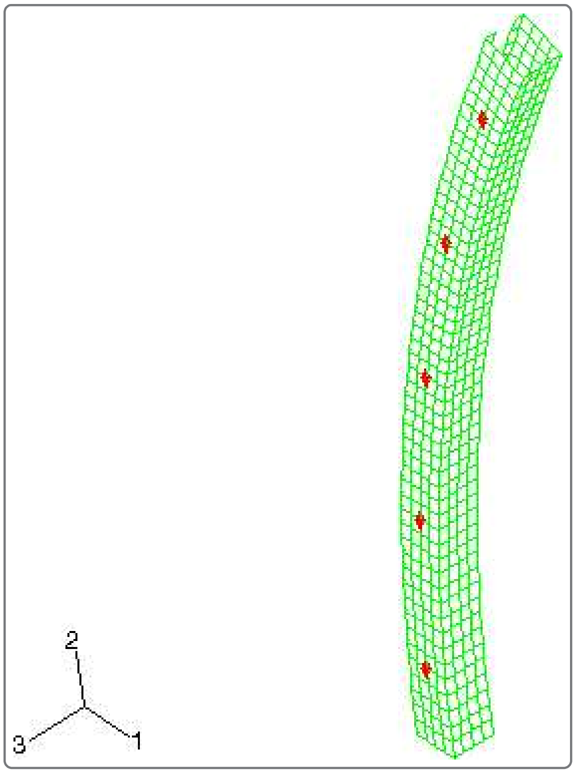

Figure 3. Deformed shape at 5.0 msec.

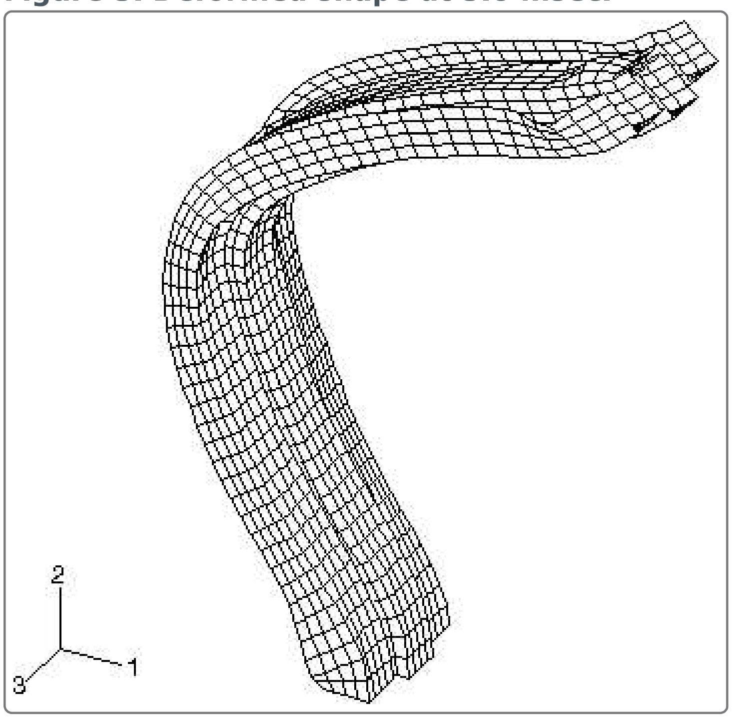

Figure 4. Deformed shape at 10.0 msec.

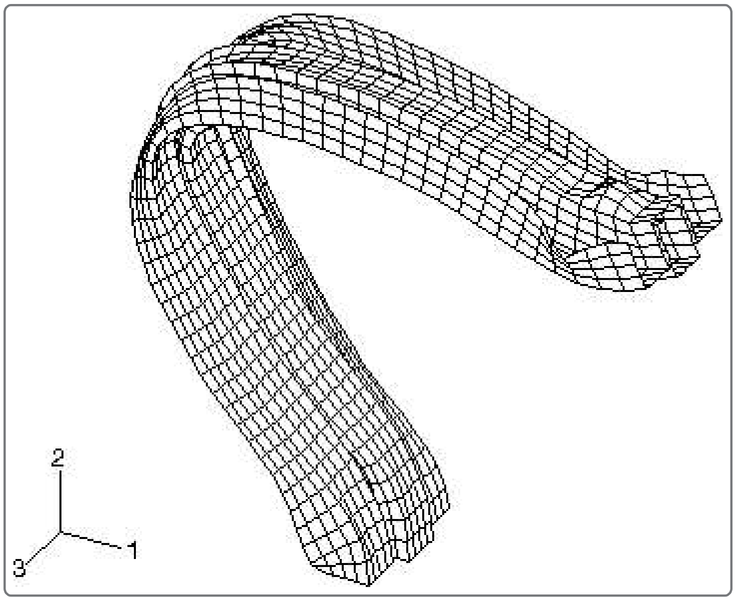

Figure 5. Connector force CTF3 in spot welds at reference nodes 25203 and 25211 for Abaqus/Standard.

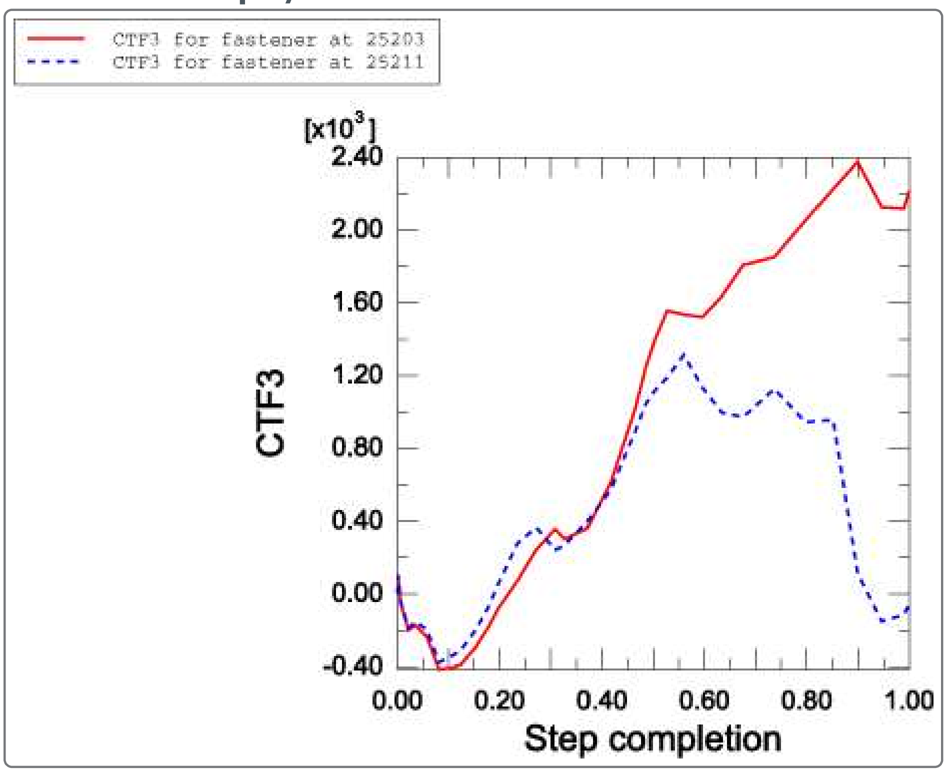

Figure 6. Connector force CTF3 in spot welds at reference nodes 25203 and 25211 for Abaqus/Explicit.

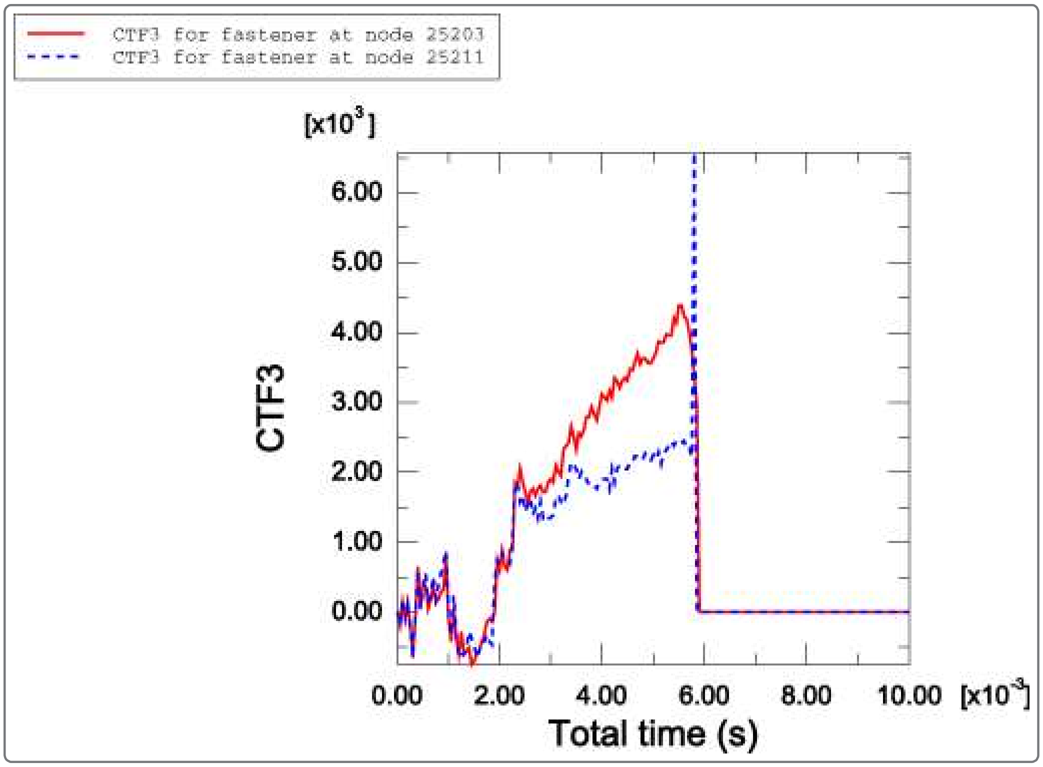

Figure 7. Time histories of the total kinetic energy, energy dissipated by friction, work done on the model, internal energy, and total energy.