Deep drawing of a square box

This example illustrates the forming of a three-dimensional shape by a deep drawing process.

The following Abaqus features are demonstrated:

- transferring results from Abaqus/Explicit to Abaqus/Standard using the import analysis technique;
- comparing results from an analysis sequence that uses Abaqus/Explicit for a forming step and Abaqus/Standard for a springback analysis with results obtained using Abaqus/Standard for both the forming and springback steps; and
- comparing characteristics of different contact formulations with finite sliding, especially with regard to the treatment of surface thickness.

This page discusses:

Products: Abaqus/Standard Abaqus/Explicit

Application description

In general, the forming procedure involves a forming step followed by a springback that occurs after the blank is removed from the tool. The goal of analyzing the forming procedure is to determine the final deformed shape after springback.

Geometry

The blank is initially square, 200 mm by 200 mm, and is 0.82 mm thick. The rigid die is a flat surface with a square hole 102.5 mm by 102.5 mm, rounded at the edges with a radius of 10 mm. The rigid square punch measures 100 mm by 100 mm and is rounded at the edges with the same 10 mm radius. The rigid blank holder can be considered a flat plate, since the blank never comes close to its edges. The geometry of these rigid parts is illustrated in Figure 1.

Materials

The blank is made of aluminum-killed steel, which is assumed to satisfy the Ramberg-Osgood relation between true stress and logarithmic strain,

$$\epsilon = (\sigma/K)^{1/n},$$

with a reference stress value (K) of 513 MPa and a work-hardening exponent (n) of 0.223. Isotropic elasticity is assumed, with a Young's modulus of 211 GPa and a Poisson's ratio of 0.3. An initial yield stress of 91.3 MPa is obtained from these data. The stress-strain behavior is defined by piecewise linear segments matching the Ramberg-Osgood curve up to a total (logarithmic) strain level of 107%, with Mises yield, isotropic hardening, and no rate dependence.

Boundary conditions and loading

Given the symmetry of the problem, it is sufficient to model only a one-eighth sector of the box. However, for easier visualization we have employed a one-quarter model. Symmetry boundary conditions are applied at the quarter edges of the blank. The punch and the blank holders are allowed to move only in the vertical direction. Allowing vertical motion of the blank holders accommodates changes in the blank thickness during forming.

Interactions

Contact interaction is considered between the blank and the punch with a friction coefficient of 0.25 and between the blank and the die with a friction coefficient of 0.125. The contact interaction between the blank and the blank holders is assumed to be frictionless.

Abaqus modeling approaches and simulation techniques

The most efficient way to analyze this type of problem is to analyze the forming step using Abaqus/Explicit and to import the results in Abaqus/Standard to analyze the springback. For verification purposes the complete analysis is also carried out with Abaqus/Standard. However, this is computationally more expensive and will be prohibitively more expensive for simulation of the forming of realistic, complex components.

This problem is used in Nagtegaal and Taylor (1991) where implicit and explicit finite element techniques for forming problems are compared. The computer time involved in running the simulation using explicit time integration with a given mesh is directly proportional to the time period of the event, since the stable time increment size is a function of the mesh size (length) and the material stiffness. Thus, it is usually desirable to run the simulation at an artificially high speed compared to the physical process. If the speed in the simulation is increased too much, the solution does not correspond to the low-speed physical problem; i.e., inertial effects begin to dominate. In a typical forming process the punch may move at speeds on the order of 1 m/sec, which is extremely slow compared to typical wave speeds in the materials to be formed (the wave speed in steel is approximately 5000 m/sec). In general, inertia forces will not play a dominant role for forming rates that are considerably higher than the nominal 1 m/sec rates found in the physical problem. Therefore, explicit solutions are obtained with punch speeds of 10, 30, and 100 m/sec for comparison with the static solution obtained with Abaqus/Standard. In the results presented here, the drawing process is simulated by moving the reference node for the punch downward through a total distance of 36 mm in 0.0036 seconds. A detailed comparison of analyses of various metal forming problems using explicit dynamic and static procedures is discussed in the paper by Nagtegaal and Taylor (1991).

Although this example does not contain rate-dependent material properties, it is common in sheet metal forming applications for this to be a consideration. If the material is rate-dependent, the velocities cannot be artificially increased without affecting the material response. Instead, the analyst can use the technique of mass scaling to adjust the effective punch velocity without altering the material properties. Rolling of thick plates contains an explanation and an example of the mass scaling technique.

Summary of analysis cases

Forming analysis with Abaqus/Explicit.	Case 1a	Using the general contact capability.
	Case 1b	Using the kinematic contact pairs.
	Case 1c	Using penalty contact pairs.
	Case 1d	Forming analysis of a fine mesh case using the general contact capability (included for the sole purpose of testing the performance of the Abaqus/Explicit code).
	Case 1e	Forming analysis of a fine mesh case using kinematic contact pairs (included for the sole purpose of testing the performance of the Abaqus/Explicit code).
Springback analysis with Abaqus/Standard.	Case 2a	Abaqus/Standard springback analysis using an import analysis with no update of the reference configuration.
	Case 2b	Abaqus/Standard springback analysis using an import analysis with update of the reference configuration.
	Case 2c	Springback analysis of a fine mesh case (included for the sole purpose of testing the performance of the Abaqus/Standard code) using an import analysis with update of the reference configuration.
Forming and springback analysis with Abaqus/Standard.	Case 3a	Using the surface-to-surface contact formulation.
	Case 3b	Using the node-to-surface contact formulation.

Analysis types

As described earlier, the import capability in Abaqus is utilized to run the forming step as an explicit dynamic analysis followed by a static stress analysis using Abaqus/Standard for calculating the springback. For comparison, results from a complete static stress analysis using Abaqus/Standard for both the forming and the springback steps are presented.

Analysis techniques

The import feature in Abaqus is used for transferring results from Abaqus/Explicit to Abaqus/Standard.

Mesh design

The blank is modeled with 4-node, bilinear finite-strain elements (type S4R); while the punch, die, and the blank holder are meshed using 4-node, three-dimensional rigid surface elements (type R3D4). The mesh design for the various parts is shown in Figure 1 and Figure 2.

Loads

The blank is held between the blank holders by applying a concentrated load of 22.87 kN. Further loading on the blank is applied by contact forces with the punch in the forming step.

Analysis steps

Using Abaqus/Explicit for the forming procedure involves a single forming step where the rigid punch is pushed against the blank while the blank is held by the blank holders by applying a concentrated load. This description applies to Cases 1a–1e. For the import analysis in Abaqus/Standard a single step is used to calculate the springback as in Cases 2a–2c. For the complete analysis in Abaqus/Standard as in Cases 3a and 3b, the following steps are adopted:

- First step: the blank holders are brought in contact with the blank by applying a small displacement to the reference point of one of the rigid blank holders.
- Second step: a concentrated load is applied to the reference point of the blank holder to hold the blank in place while maintaining contact.
- Following steps: the forming is effected by pushing the rigid punch against the blank.
- Final two steps: the springback is analyzed by deactivating the contact pairs.

Output requests

The output variables STH for shell thickness and PEEQ for equivalent plastic strain are specifically requested along with preselected variables. Further, the history of reaction force and displacement for the punch is also requested.

Case 1a: Explicit forming analysis using general contact

This analysis pertains only to the forming step. For the complete analysis the forming step in this case needs to be followed by a springback analysis (either Case 2a or Case 2b).

Interactions

General contact is used (see the general contact specification) to define contact interactions in this case. This allows very simple definitions of contact with very few restrictions on the types of surfaces involved (see About General Contact in Abaqus/Explicit). However, general contact does not account for changes in shell thickness by default. Consequently, the general contact surface property assignment must account for thinning of the blank.

Case 1b: Explicit forming analysis using kinematic contact pairs

This analysis again pertains only to the forming step. For the complete analysis the forming step needs to be followed by a springback analysis (either Case 2a or Case 2b).

Interactions

Contact pairs are defined to include blank interaction with the punch, die, and the blank holder separately with appropriate friction behavior as previously specified. The contact pair algorithm, which is specified in the contact pair definition, has more restrictions on the types of surfaces involved and often requires more careful definition of contact (see About Contact Pairs in Abaqus/Explicit). Contact interactions are defined between all element-based surfaces in the model.

Case 1c: Explicit forming analysis using penalty contact pairs

This analysis pertains only to the forming step. The springback calculations have to be done separately (Case 2a or Case 2b).

Interactions

Penalty contact is specified for contact pairs to include blank interaction with the punch, die, and the blank holder separately with appropriate friction behavior.

Case 1d: Explicit forming analysis using general contact with a refined mesh

In this case the mesh for the blank is uniformly refined so that the number of elements in each direction is twice the number in the previous cases. This case is run to purely benchmark the efficiency of performing an explicit analysis.

Interactions

The contact interactions are exactly the same as in Case 1a.

Case 1e: Explicit forming analysis using kinematic contact pairs with a refined mesh

In this case the refined mesh defined in Case 1d is utilized for performing the explicit forming analysis.

Interactions

The contact interactions are exactly the same as in Case 1b.

Case 2a: Static springback analysis with no update of the reference configuration during import

For running this case, a prior explicit forming analysis (Case 1a, Case 1b, or Case 1c) should have been completed for importing results into Abaqus/Standard. By specifying an import analysis with no update of the reference configuration, the displacements are the total values relative to the original reference configuration before the forming analysis. This makes it easy to compare the results with the analysis in which both the forming and springback are analyzed with Abaqus/Standard.

Boundary conditions

Boundary conditions are imposed in the Abaqus/Standard analysis to prevent rigid body motion and for symmetry. The node at the center of the box is fixed in the z-direction.

Interactions

No contact interactions are used in this analysis once the deformed sheet with its material state at the end of Abaqus/Explicit is imported.

Case 2b: Static springback analysis with update of the reference configuration during import

Similar to Case 2a, a prior explicit forming analysis (Case 1a, Case 1b, or Case 1c) should have been completed for importing results into Abaqus/Standard. However, specifying an import analysis with update of the reference configuration implies that the displacements are relative to the deformed configuration at the end of the forming analysis. The boundary conditions and interactions are exactly the same as Case 2a.

Case 2c: Static springback analysis using a refined mesh with update of the reference configuration during import

For running this case, Case 1d or Case 1e for explicit forming analysis should have been completed for importing results into Abaqus/Standard. Here again, specifying an import analysis with update of the reference configuration implies that the displacements are relative to the deformed configuration at the end of the forming analysis. The boundary conditions and interactions are exactly the same as Case 2a.

Case 3a: Static analysis of forming and springback using surface-to-surface contact

In this analysis both the forming and the springback steps are analyzed in Abaqus/Standard.

Interactions

In this case the surface-to-surface contact formulation is invoked. Since double-sided surfaces are not available in Abaqus/Standard, two single-sided surfaces are used to model the blank when the forming step is modeled in Abaqus/Standard: one surface to model the top of the blank and one to model the bottom of the blank. The surface-to-surface contact formulation considers the original shell thickness by default throughout the analysis. There is no option to consider the current shell thickness instead of the original shell thickness.

Solution controls

Contact stabilization is used to avoid chattering between the blank and the rigid surfaces it is in contact with. In addition, the adaptive automatic stabilization scheme is applied to improve the robustness of the static analysis.

Case 3b: Static analysis of forming and springback using node-to-surface contact

As in Case 3a, both the forming and the springback steps are analyzed in Abaqus/Standard.

Interactions

In this case the node-to-surface contact formulation is used. Since, shell thickness cannot be considered by node-to-surface finite-sliding contact, “softened” contact is used to approximate the thickness (see the modified contact pressure-overclosure relationship).

Discussion of results and comparison of cases

Figure 3, Figure 5, and Figure 4 show contours of shell thickness in the blank at the end of the forming step before springback in Abaqus/Explicit (Case 1a) and Abaqus/Standard analyses (Case 3a and Case 3b), respectively. Figure 6, Figure 7, and Figure 8 show contours of equivalent plastic strain in the blank in the final deformed shape for the Abaqus/Explicit and the two Abaqus/Standard analyses, respectively. The predicted results are very similar. The Abaqus/Explicit results match the surface-to-surface contact formulation in Abaqus/Standard more closely than the node-to-surface results in Abaqus/Standard. This observation is true for both the equivalent plastic strain contours and shell thickness contours and is a consequence of the intrinsic differences between the various contact formulations. The node-to-surface formulation in Abaqus/Standard accounts for the shell thickness indirectly by using carefully specified pressure-overclosure relationships (soft contact). The other analyses use contact formulations that account for shell thickness directly. Despite the fact that the surface-to-surface formulation in Abaqus/Standard uses the original shell thickness throughout the analysis, the results correlate well.

Closer inspection of the results reveals that the corners of the box are formed by stretching, whereas the sides are formed by drawing action. This effect leads to the formation of shear bands that run diagonally across the sides of the box, resulting in a nonhomogeneous wall thickness. The material draws unevenly from the originally straight sides of the blank. Applying a more localized restraint near the midedges of the box (for example, by applying drawbeads) and relaxing the restraint near the corners of the box is expected to increase the quality of the formed product.

Figure 9 shows the reaction force on the punch, and Figure 10 shows the thinning of an element at the corner of the box. Here again, the results from the surface-to-surface formulation in Abaqus/Standard match those from Abaqus/Explicit better than the node-to-surface contact formulation in Abaqus/Standard. Despite the approximate treatment of surface thickness via the pressure-overclosure relationship for the node-to-surface formulation, the shell thicknesses predicted by Abaqus/Explicit and the node-to-surface formulation in Abaqus/Standard differ only by about 4%, reflecting the overall quality of the results.

The springback analysis runs in 6 increments for both of the contact formulations in Abaqus/Standard. Most of the springback occurs in the *z*-direction, and the springback is not significant. The corner of the outside edge of the formed box drops approximately 0.35 mm, while the vertical side of the box rises by approximately 0.26 mm. Figure 11 shows a contour plot of the displacements in the *z*-direction obtained from the springback analysis using the node-to-surface formulation.

The analysis with no reference configuration update yields similar results. However, in this case the displacements are interpreted as total values relative to the original configuration.

Files

Case 1a: Explicit forming analysis using general contact

deepdrawbox_exp_form.inp

Input file for the explicit forming step.

Case 1b: Explicit forming analysis using kinematic contact pairs

deepdrawbox_exp_form_cpair.inp

Input file for the explicit forming step.

Case 1c: Explicit forming analysis using penalty contact pairs

deepdrawbox_exp_form_plty_cpair.inp

Input file for the explicit forming step.

Case 1d: Explicit forming analysis using general contact with a refined mesh

deepdrawbox_exp_finemesh.inp

Input file for the explicit forming step.

Case 1e: Explicit forming analysis using kinematic contact pairs with a refined mesh

deepdrawbox_exp_finemesh_cpair.inp

Input file for the explicit forming step.

Case 2a: Static springback analysis with UPDATE=NO during import

deepdrawbox_std_importno.inp

Input file for the static springback step.

Case 2b: Static springback analysis with UPDATE=YES during import

deepdrawbox_std_importyes.inp

Input file for the static springback step.

Case 2c: Static springback analysis using a refined mesh with UPDATE=YES during import

deepdrawbox_std_finesprngback.inp

Input file for the static springback step with a refined mesh for the blank.

Case 3a: Static analysis of forming and springback using surface-to-surface contact

deepdrawbox_std_both_surf.inp

Input file for the complete static analysis.

deepdrawbox_std_both_surf_stabil_adap.inp

Input file for the complete static analysis with adaptive stabilization.

Case 3b: Static analysis of forming and springback using node-to-surface contact

deepdrawbox_std_both.inp

Input file for the complete static analysis.

References

Nagtegaal J. C. and L. M. Taylor, "Comparison of Implicit and Explicit Finite Element Methods for Analysis of Sheet Forming Problems,"VDI Berichte No. 894, 1991.

Figures

Figure 1. Meshes for the die, punch, and blank holder.

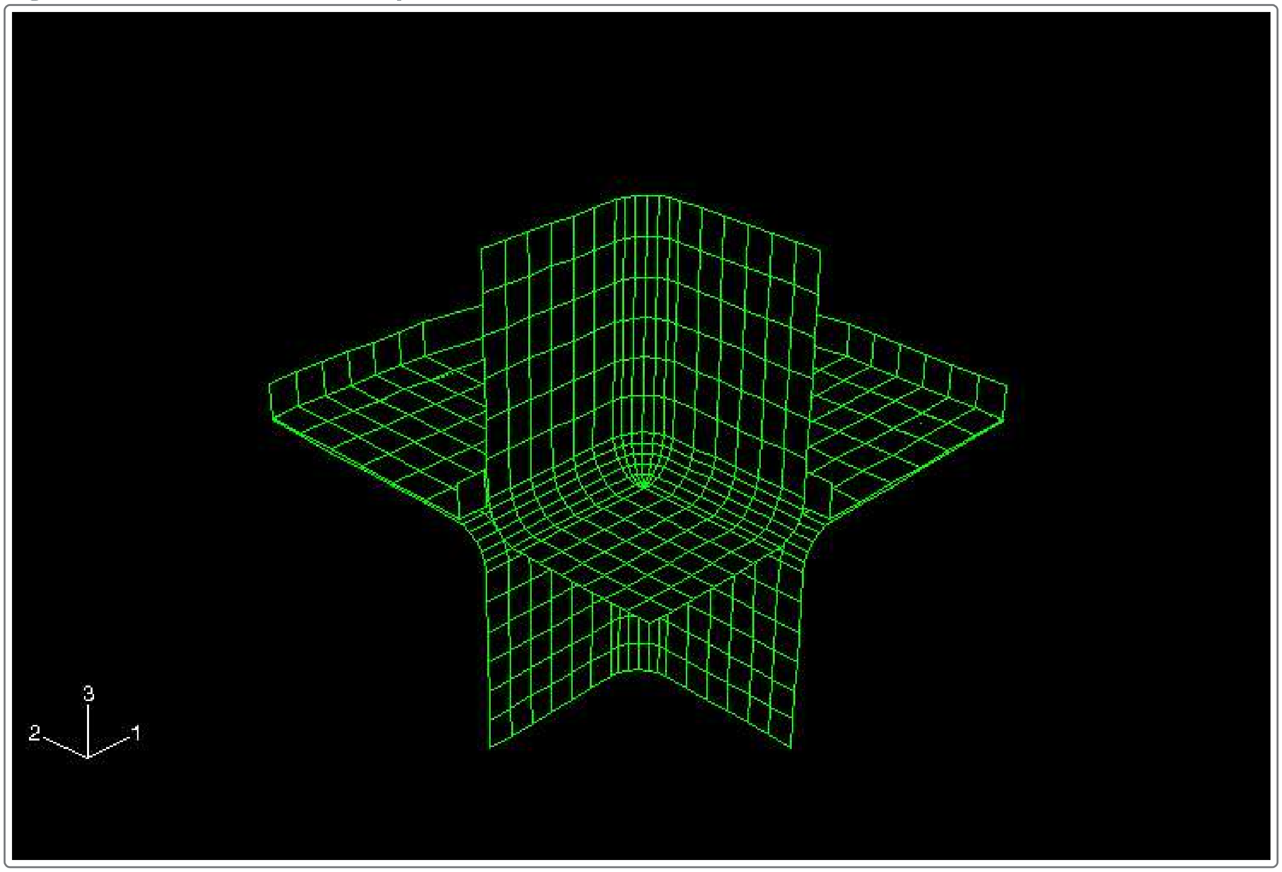

Figure 2. Undeformed mesh for the blank.

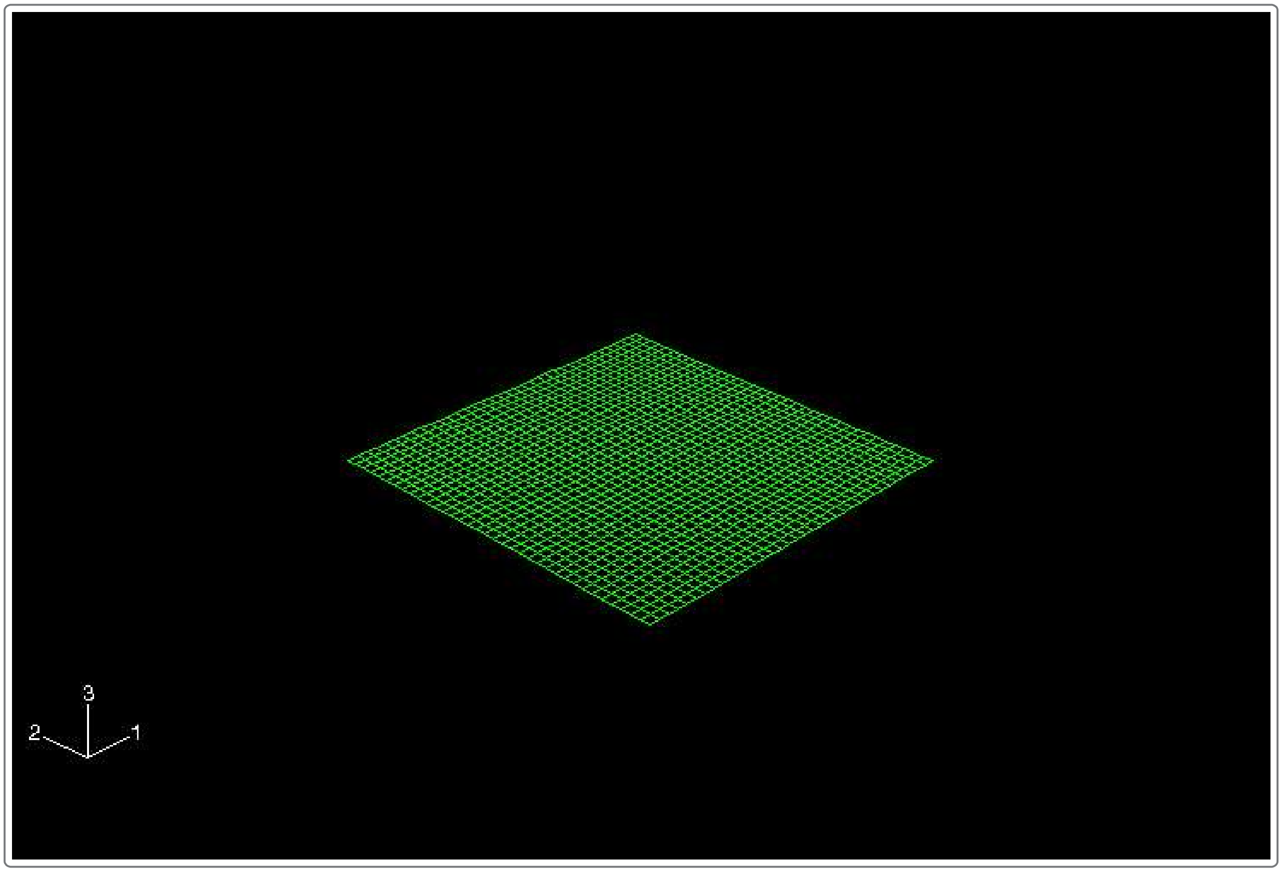

Figure 3. Contours of shell thickness with Abaqus/Explicit.

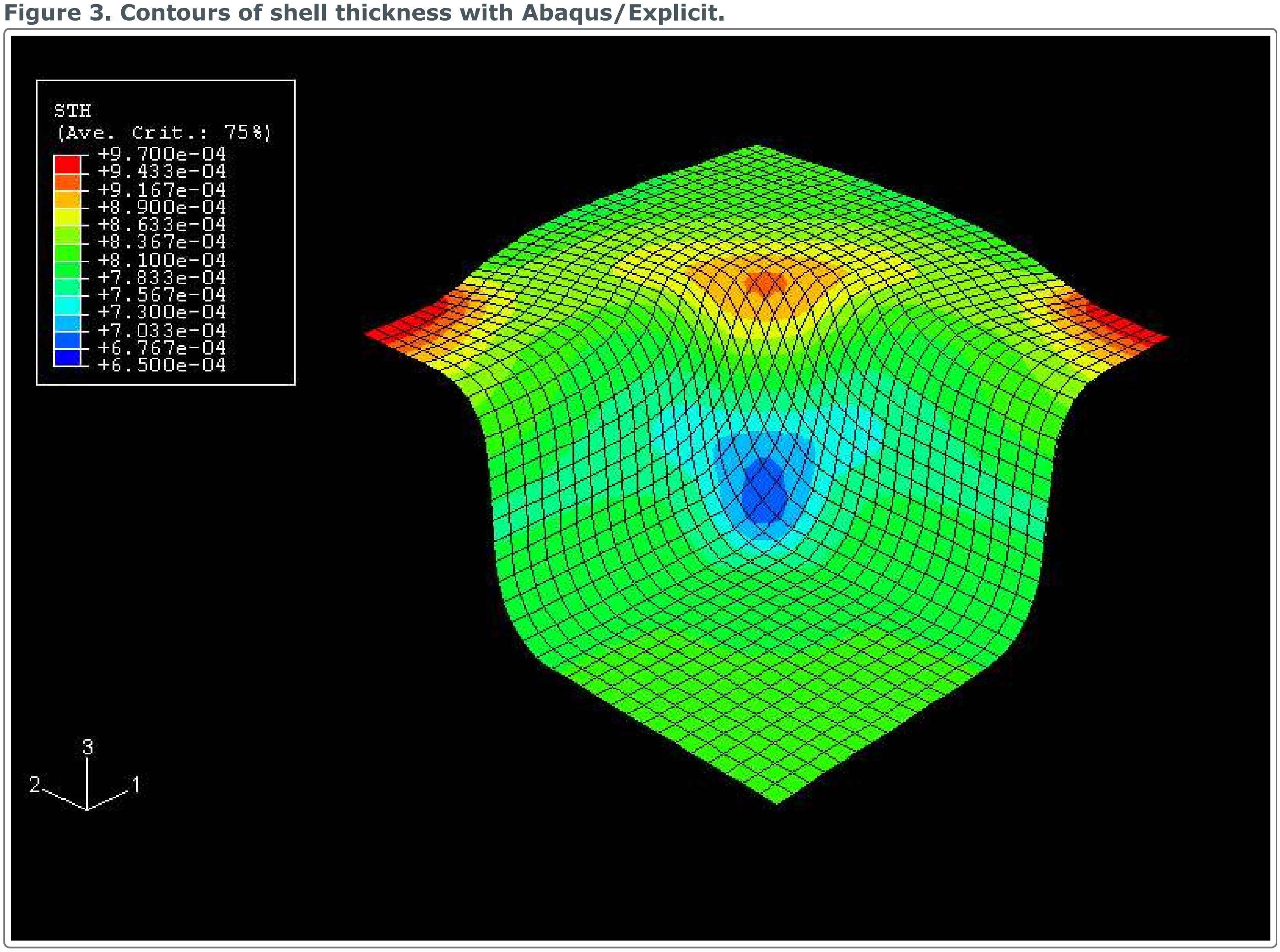

Figure 4. Contours of shell thickness with Abaqus/Standard using surface-to-surface contact formulation.

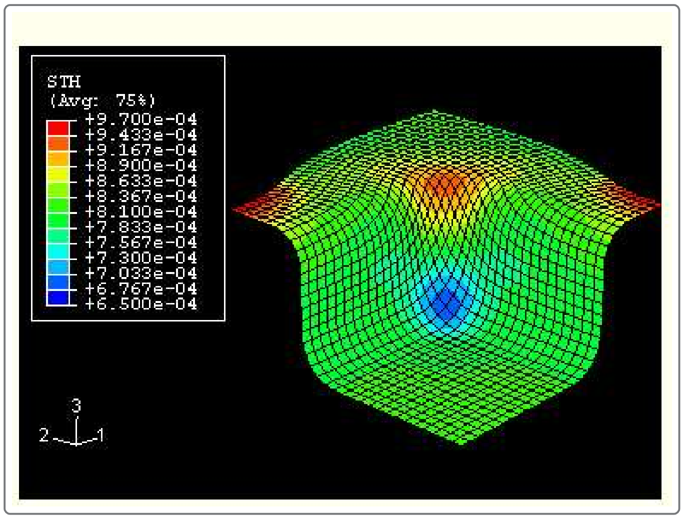

Figure 5. Contours of shell thickness with Abaqus/Standard using node-to-surface contact formulation.

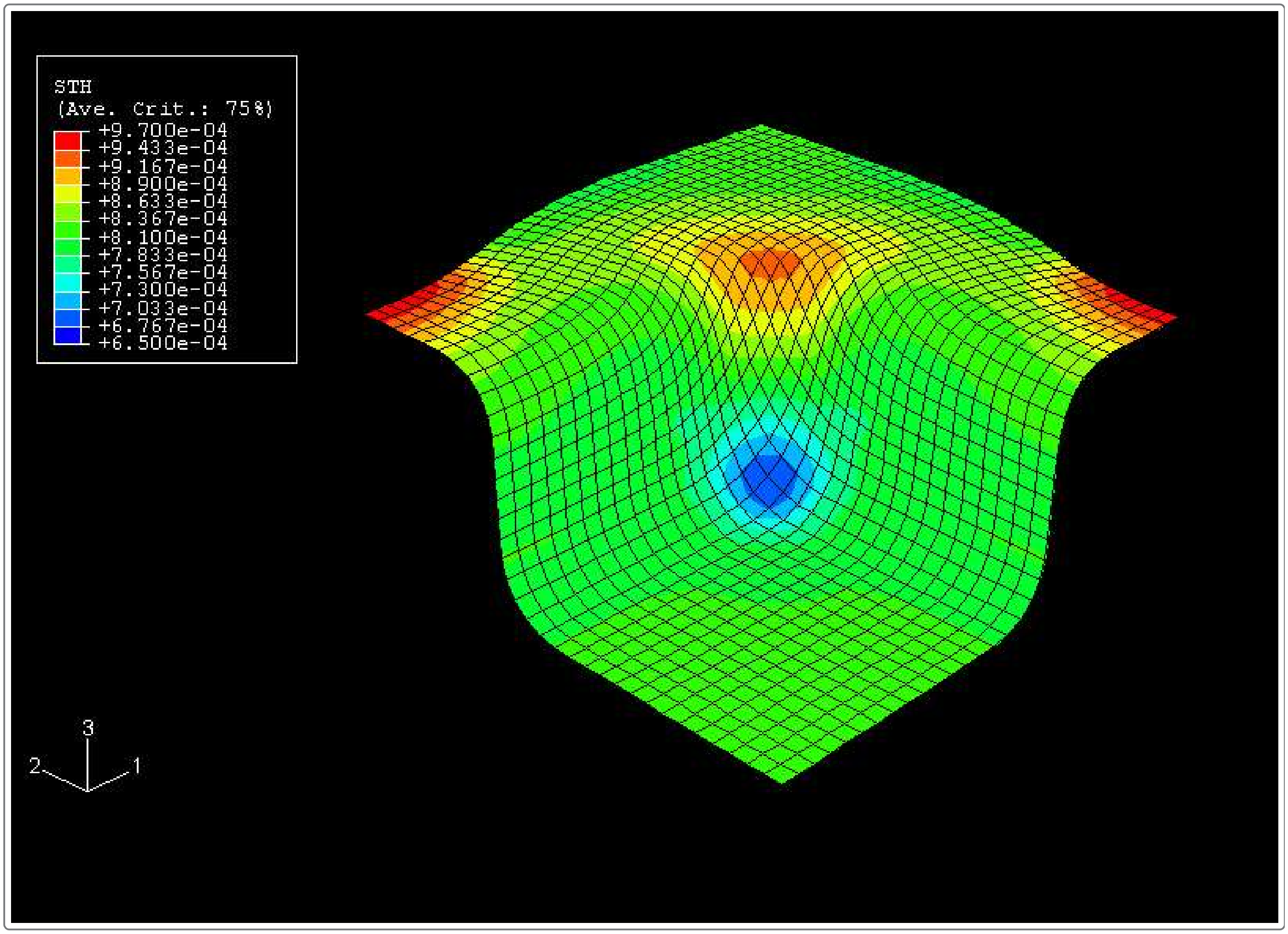

Figure 6. Contours of equivalent plastic strain with Abaqus/Explicit.

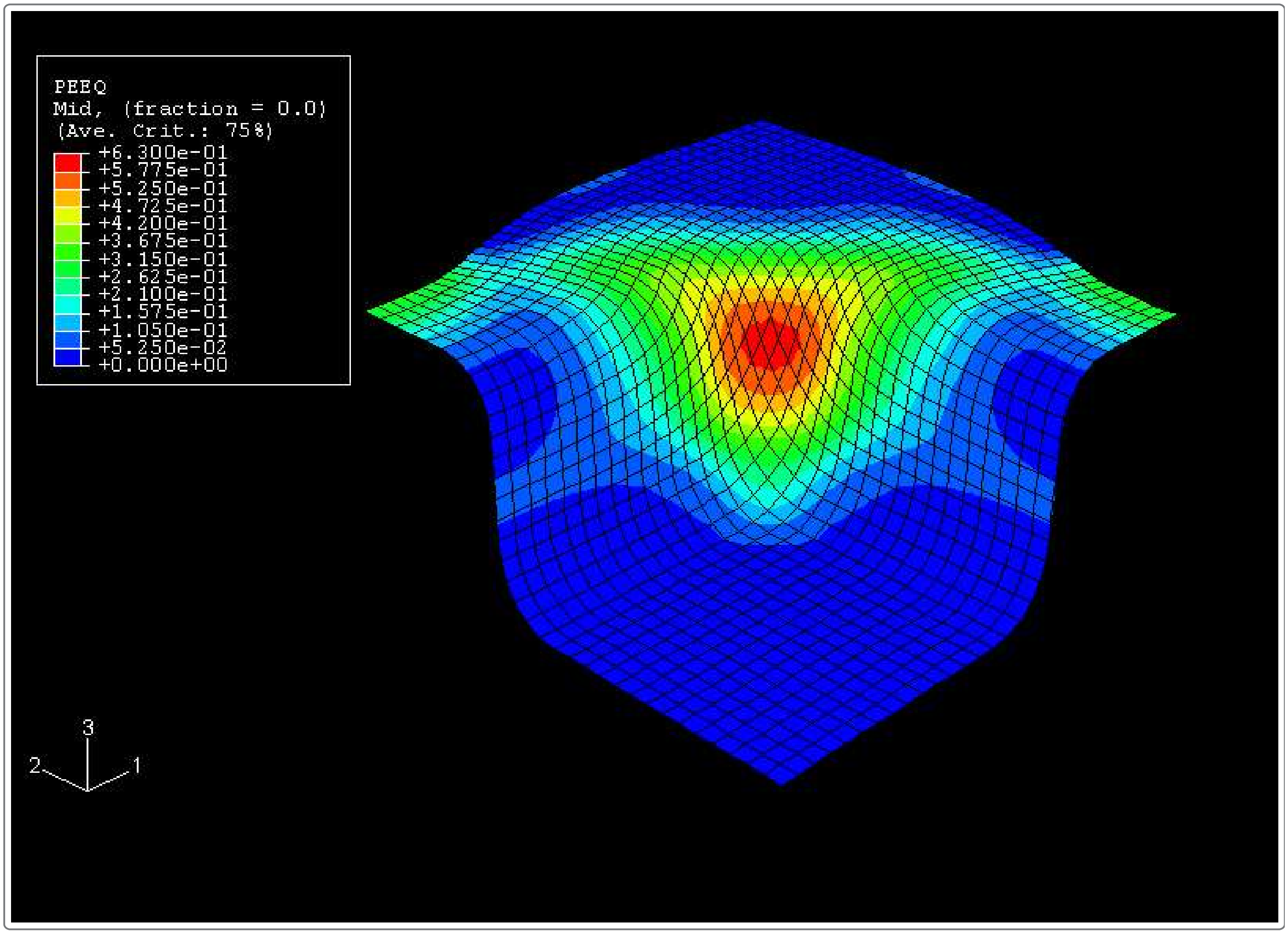

Figure 7. Contours of equivalent plastic strain with Abaqus/Standard using surface-to-surface contact formulation.

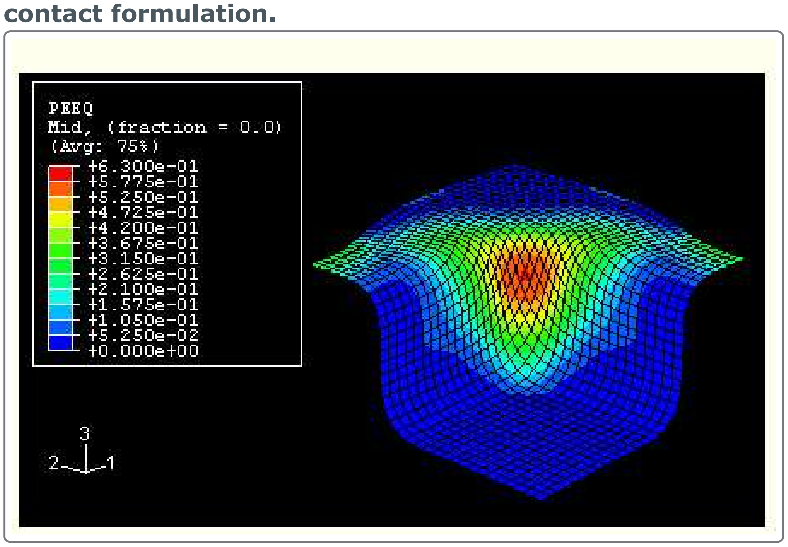

Figure 8. Contours of equivalent plastic strain with Abaqus/Standard using node-to-surface contact formulation.

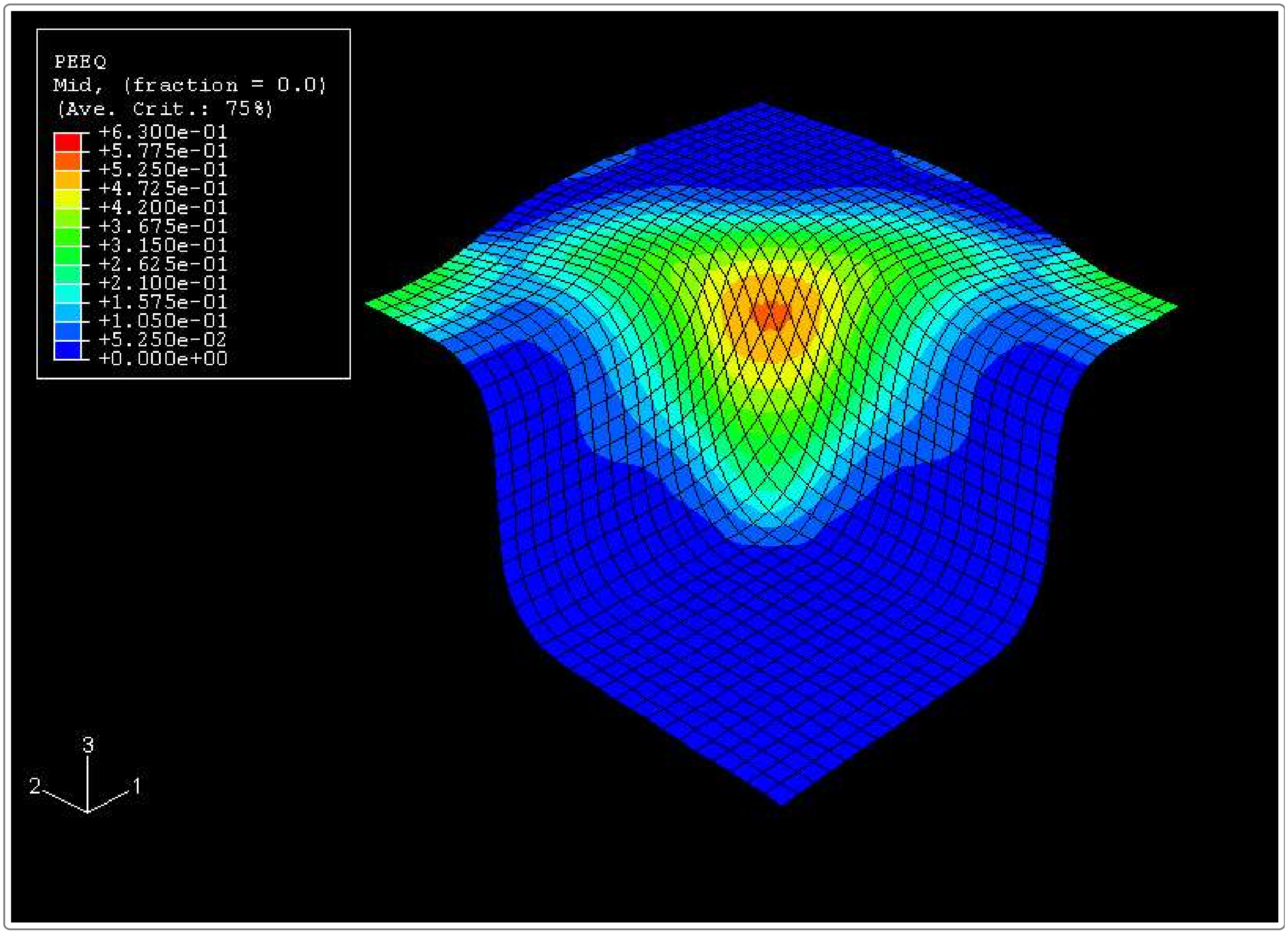

Figure 9. Reaction force on the punch versus punch displacement.

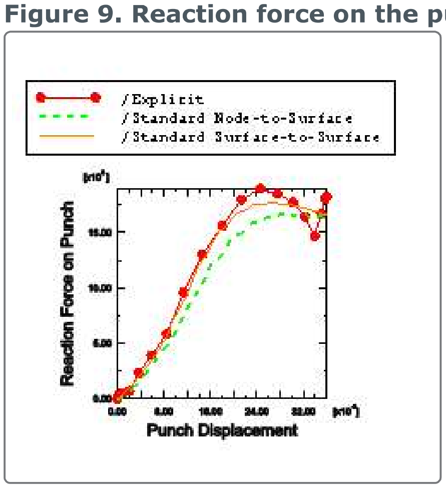

Figure 10. Shell thickness of the thinnest part of the blank versus time.

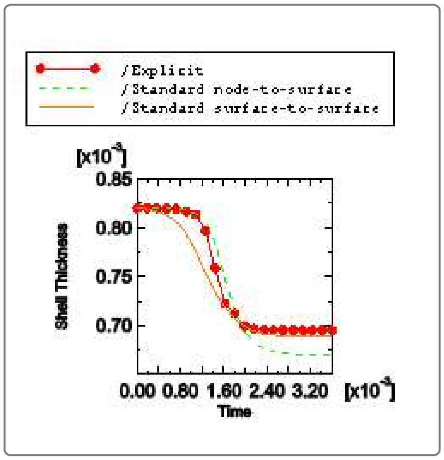

Figure 11. Contour plot showing the springback in the *z*-direction.