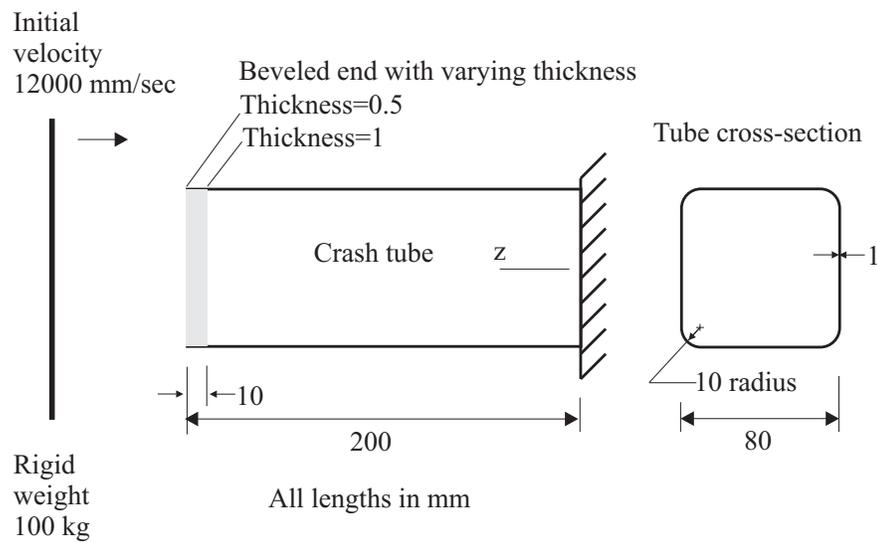


Problem description

Consider the crushing of a crash tube by a rigid weight:



Properties of rigid weight:

X, Y, Z mass
= 0.1 N-s²/mm
X, Y moment of inertia
= 2296 N-s²-mm
Z moment of inertia
= 427 N-s²-mm

(corresponds to a steel block with
dimensions 160×160×500)

Material properties,
plastic-cyclic material model:

$E=207000$ MPa
 $\nu=0.3$
 $\rho=7850$ kg/m³ = 7.85×10^{-9} N-s²/mm⁴
 $\sigma_y=225$ MPa

Nonlinear kinematic hardening:
 $h=280000$ MPa
 $\zeta=1300$

Contact conditions: Friction coefficient = 0.2 between weight and tube,
also for tube self-contact

We want to compute the force-deflection curve for the tube. We will compute this curve using both an implicit and an explicit analysis.

Problem 52: Crushing of a crash tube

In this problem solution, we will demonstrate the following topics that have not been presented in previous problems:

- Defining shell elements with varying thickness
- Using the plastic-cyclic material model
- Using double-sided contact
- Using the Noh-Bathe method of explicit time integration
- Using the penalty contact algorithm

Before you begin

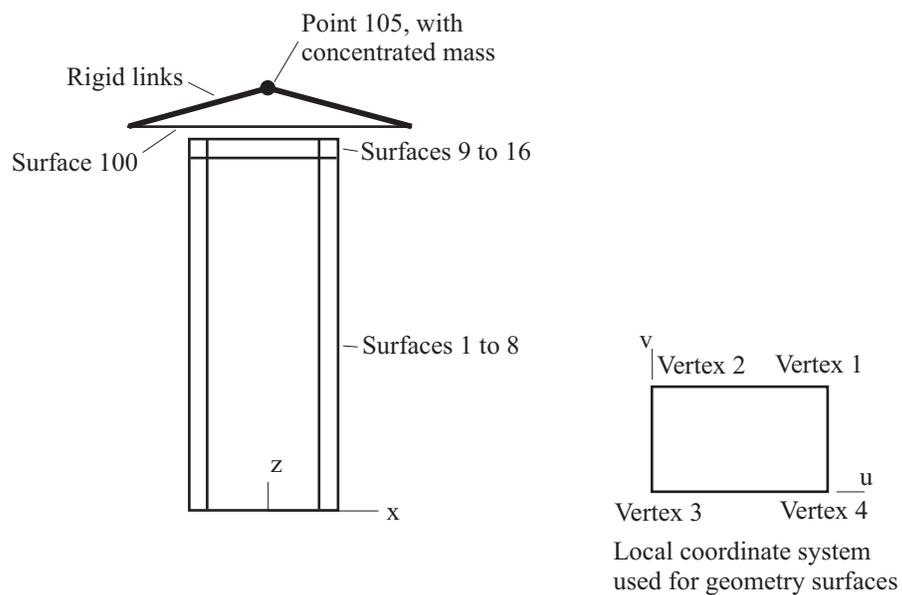
Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem can be solved with the 900 nodes version of the ADINA System.

Much of the input for this problem is stored in the following files: prob52_1.in, prob52_1.pl0, prob52_2.pl0. You need to copy these files from the folder samples\primer into a working directory or folder before beginning this analysis.

Overview of model definition

The figure below shows the geometry used to model the crush tube:



The beveled end of the tube is (approximately) modeled by reducing the shell thicknesses at the end of the tube. In order to reduce the shell thicknesses, it is necessary to know the orientation of the geometry surfaces coordinate system. The coordinate system of the geometry surfaces is shown in the figure above.

Two contact groups are used. Contact group 1 models the contact between the rigid surface and the tube, and contact group 2 models the self-contact of the tube.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Implicit time integration

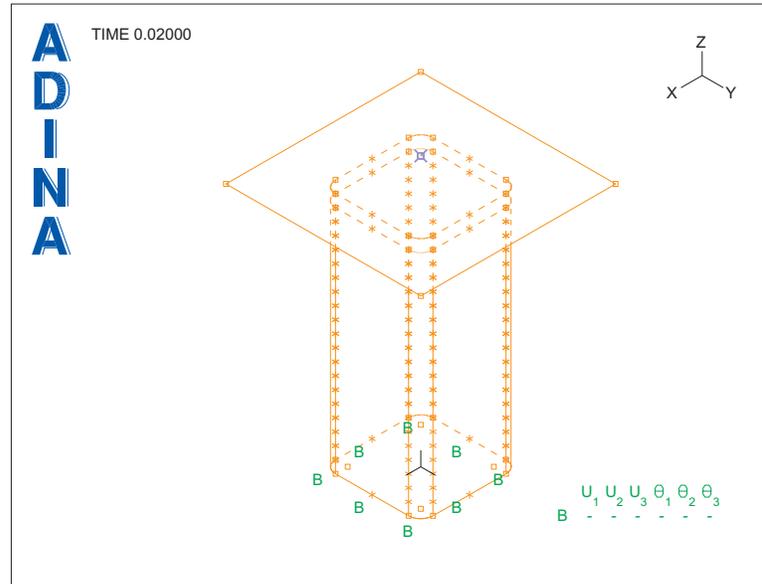
Model definition

We have prepared a batch file (`prob52_1.in`) that defines the following items:

- ▶ Problem heading
- ▶ Control data, including solution tolerances. Note that all shell nodes are assigned local rotational degrees of freedom (`MASTER ... SHELLNDOF=5`).
- ▶ Time stepping. 200 time steps of size $1E-4$ are used.
- ▶ Geometry points, lines, surfaces
- ▶ Thickness for geometry surfaces 1 to 16, except for the deviations (see below)
- ▶ Boundary conditions
- ▶ Rigid links connecting geometry surface 101 to geometry point 105
- ▶ Element group 1, containing soft springs attached to all the degrees of freedom of geometry point 5. These springs prevent the degrees of freedom in element group 5 from being deleted at the time of data file generation.
- ▶ Subdivision data for the surfaces

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob52_1.in` and click Open. The graphics window should look something like the figure on the next page.

Problem 52: Crushing of a crash tube



Surface thicknesses: The 8 geometry surfaces close to the top of the tube will have a varying thickness applied to them. Choose Geometry→Surfaces→Thickness and notice that all 16 surfaces have a thickness of 1.0, with no deviations. For surfaces 9 to 16, set both Deviation 1 and Deviation 2 to -0.5 and click OK.

Plastic-cyclic material model: Click the Manage Materials icon **M** and click the Cyclic button. Add material 1, and click the ... button to the right of the Isotropic Hardening Rule field. In the Define Isotropic Hardening Rule dialog box, add rule 1, make sure that the Type is Bilinear, set the Yield Stress to 225 and click OK. In the Define Plastic-Cyclic Material dialog box, set the Isotropic Hardening Rule to 1. Now click the ... button to the right of the Kinematic Hardening Rule field. In the Define Kinematic Hardening Rule dialog box, add rule 1, then, in the first row of the table, set the "Linear Constant h" to 280000 and the "Nonlinear Constant zeta" to 1300, and click OK. In the Define Plastic-Cyclic Material dialog box, set the Kinematic Hardening Rule to 1. Now set the Young's Modulus to 207000, the Poisson's Ratio to 0.3, the Density to 7.85E-9, then click OK. Click Close to close the Manage Material Definitions dialog box.

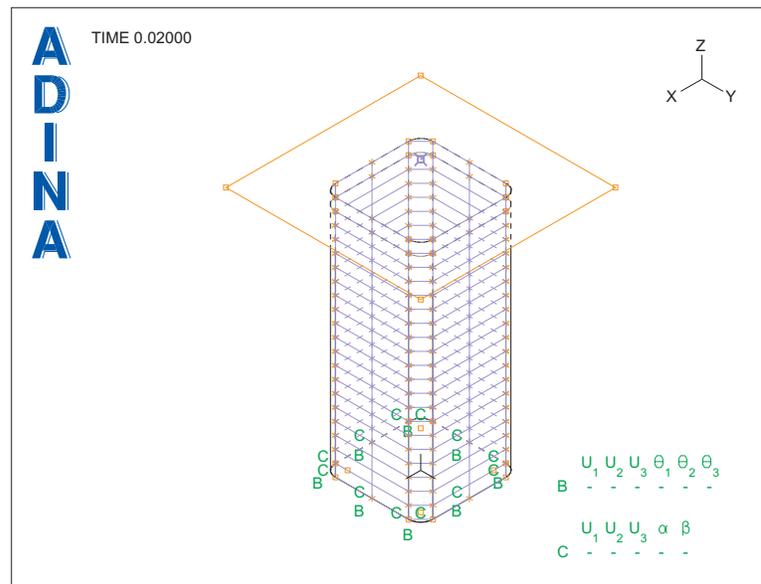
Concentrated mass: Choose Model→ Concentrated Masses, edit the table as follows, then click OK:

Point	X- Translation	Y- Translation	Z- Translation	X- Rotation	Y- Rotation	Z- Rotation
105	0.1	0.1	0.1	2296	2296	427

Initial conditions: Choose Model→Initial Conditions→Define, add initial condition V, and, in the first row of the table, set the Variable to Z-VELOCITY, the Value to -12000 and click Save. Now click the Apply... button, and, in the Apply Initial Conditions dialog box, set the Point to 105 in the first row of the table, then click OK twice to close both dialog boxes.

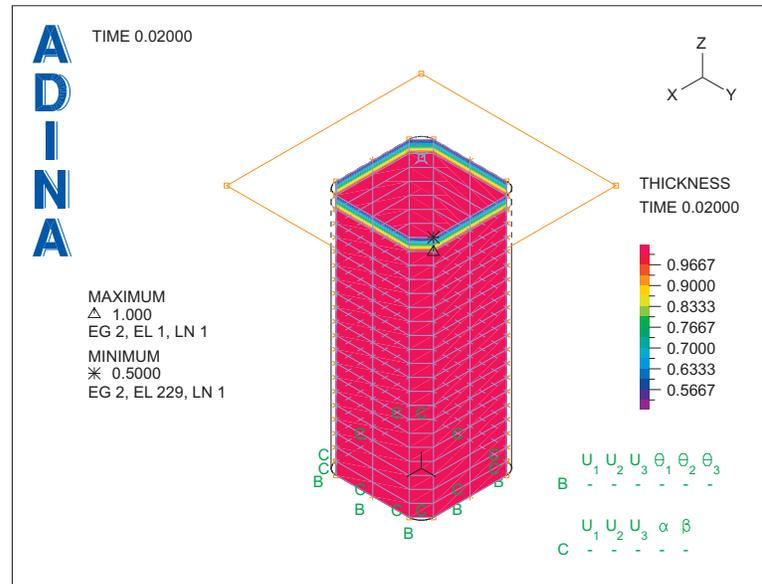
Element definition: Click the Element Groups icon , add group 2, set the Type to Shell, set the Integration Type to Trapezoidal and the Integration Order to 3. Then click the 3D-Shell tab, check the Use 3D-Shell button and click OK. Now click the Mesh Surfaces icon , click the Auto... button, set From to 1 and To to 16, then click OK. The Surface table should be filled in with numbers 1 to 16. Click OK to close the Mesh Surfaces dialog box.

The graphics window should look something like this:



Let's check the thicknesses. Click the Create Band Plot icon , set the Variable to (Thickness:THICKNESS) and click OK. The graphics window should look something like the figure on the next page.

Problem 52: Crushing of a crash tube



Click the Clear Band Plot icon  to remove the band plot.

Contact group 1: Click the Contact Groups icon , add group 1 and set the Type to 3-D Contact. Set the Compliance Factor to 1E-4 and the Contact Surface Offset to None. Now click the Advanced tab, set the "Friction Contact v-Function Parameter" to 100 and click OK.

Contact surfaces for contact group 1: Click the Define Contact Surfaces icon  and add contact surface 1. Check the "Specify Orientation" button, and, in the first row of the table, set the Surf/Face to 101, the Orientation to "Opposite to Geometry" and click Save. Add contact surface 2, click the Auto... button, set From to 1 and To to 16, then click OK. The table should be filled in with numbers 1 to 16. Click OK to close the Define Contact Surface on Geometry dialog box.

We need to generate a contact segment on contact surface 1. Click the Mesh Rigid Contact Surface icon , set the Contact Surface to 1, the Number of Nodes per Segment to 4 and click OK.

Contact pair for contact group 1: Click the Define Contact Pairs icon , add contact pair 1, set the Contactor Surface to 2, the Coulomb Friction Coefficient to 0.2 and click OK.

Contact group 2: Click the Contact Groups icon  and add group 2. Set the Compliance Factor to 1E-3, set the Contact Surface Action to Double Side and the Contact Surface Offset to None. Now click the Advanced tab, set the "Friction Contact v-Function Parameter" to 100 and click OK.

Contact surfaces for contact group 2: Click the Define Contact Surfaces icon  and add contact surface 1. Click the Auto... button, set From to 1 and To to 16, then click OK. The table should be filled in with numbers 1 to 16. Click OK to close the Define Contact Surface on Geometry dialog box.

Contact pair for contact group 2: Click the Define Contact Pairs icon , add contact pair 1, set the Coulomb Friction Coefficient to 0.2 and click OK.

Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

Click the Save icon  and save the database to file prob52. Click the Data File/Solution icon , set the file name to prob52_im, make sure that the Run Solution button is checked and click Save.

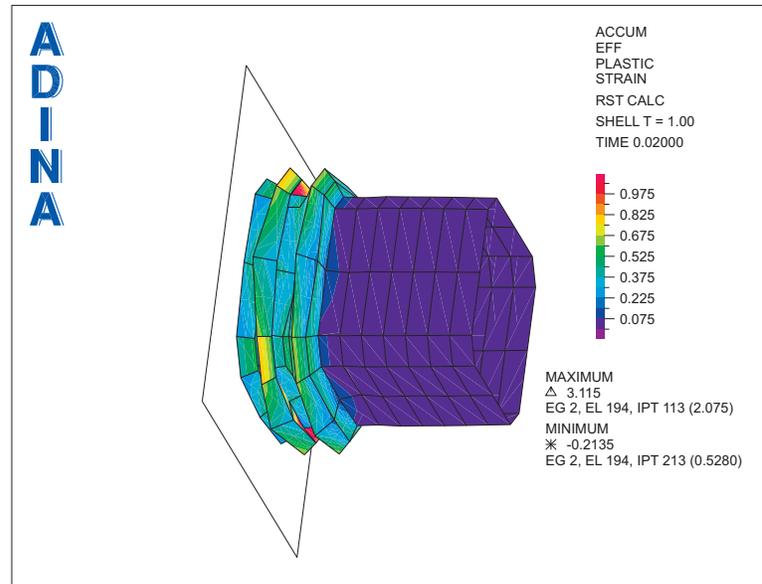
ADINA Structures runs for 200 steps.

When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob52_im.

Post-processing

Deformed mesh: We have put the commands for plotting the tube with accumulated effective plastic strains in a batch file (prob52_1.p1o). Choose File→Open Batch, navigate to the working directory or folder, select the file prob52_1.p1o and click Open. The AUI processes the commands in the batch file. The graphics window should look something like the figure on the next page.

Problem 52: Crushing of a crash tube

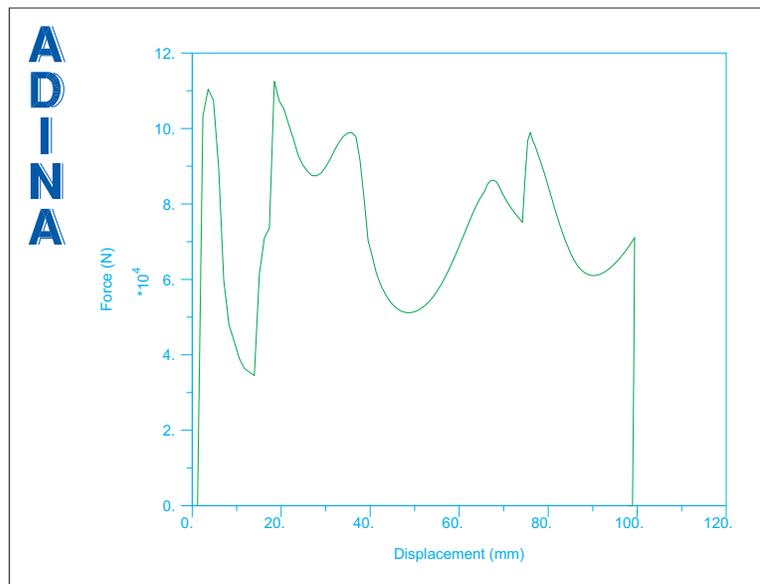
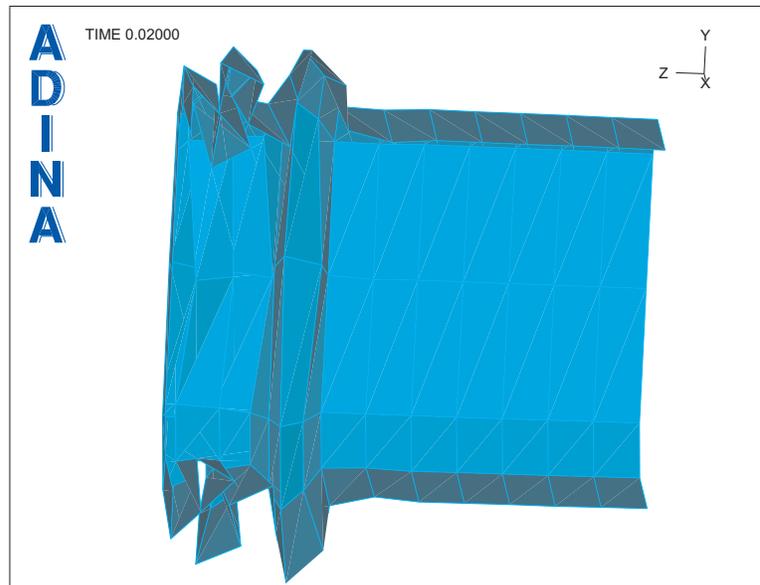


Click the Movie Load Step icon  to create an animation of the tube crushing. When the animation is finished, click the Animate icon , or choose Display→Animate, to view the animation. Notice that the tube absorbs all of the kinetic energy of the rigid weight, and that the rigid weight separates from the tube at the end of the analysis.

Now click the Clear icon , click the Mesh Plot icon , click the Cut Surface icon , set the Type to Cutting Plane, uncheck the Display the Plane(s) button, set "Below the Cutplane" to "Display as Usual", "Above the Cutplane" to "Do not Display" and click OK. In the Model Tree, expand the Zone entry, right-click the EG2 field and choose Display, then click the Shading icon . Use the mouse to rearrange the graphics window until the graphics window looks something like the top figure on the next page.

The plot shows the self-contact of the tube.

Force-deflection curve: We have put the commands for plotting the force-deflection curve in a batch file (prob52_2.p1o). Choose File→Open Batch, navigate to the working directory or folder, select the file prob52_2.p1o and click Open. The AUI processes the commands in the batch file. The graphics window should look something like the bottom figure on the next page.



Choose Graph→List and scroll to see the first solution time for which the force returns to zero. This should be time 1.82000E-2, with corresponding displacement 9.89701E+01 (mm).

Explicit analysis

Now we will repeat the analysis, using explicit time integration. Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file prob52.ldb from the recent file list near the bottom of the File menu.

Heading: Choose Control→Heading, set the Heading to "Primer problem 52: Crushing of a tube, explicit" and click OK.

Explicit analysis: Set the Analysis Type to Dynamics-Explicit and click the Analysis Options icon . Set the Method to "Noh-Bathe", make sure that the Time Step is set to "Automatic (Use Total Time Specified)", set the Time Step Magnitude Scaling Factor to 0.7 and click OK.

Plastic-cyclic material model: Click the Manage Materials icon  and click the Cyclic button. Set the Stress Integration Factor (beta) to 1.0 and click OK, then Close, to close both dialog boxes.

Contact algorithm: Click the Contact Control icon , set the Default Contact Algorithm to Penalty and click OK.

Contact groups: Click the Contact Groups icon  and choose group 1. Click the Advanced tab, and, in the Penalty Algorithm Stiffness box, set the Normal Stiffness to "Use Specified Value", and the value to 5000, and also set the Tangential Stiffness to "Use Specified Value", and the value to 5000. Click Save, then choose group 2, click the Advanced tab, and set the Normal and Tangential Stiffness to 5000 in the same way. Click OK to close the dialog box.

Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file

Click the Save icon  and save the database to file prob52. Click the Data File/Solution icon , set the file name to prob52_ex, make sure that the Run Solution button is checked and click Save.

ADINA Structures runs for about 12000 steps. When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file prob52_ex.

Post-processing

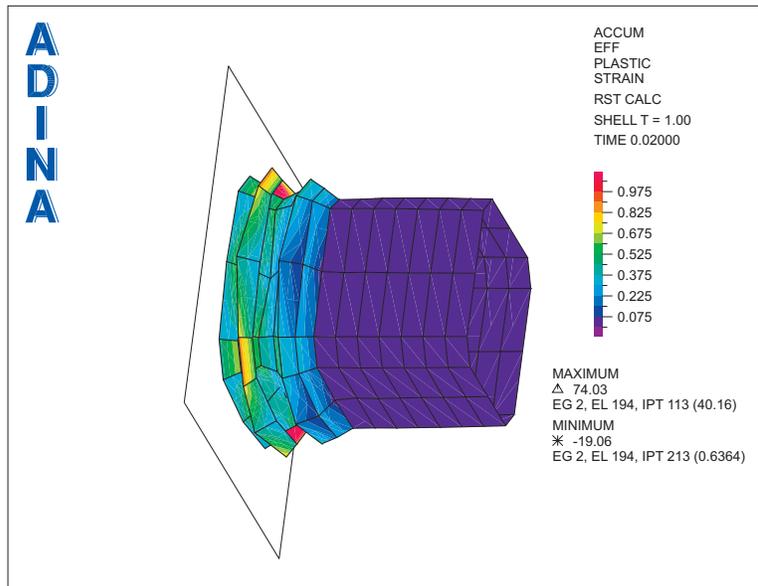
Follow the instructions given above to post-process the model. We obtain the results shown on the next page.

Choose Graph→List and scroll to see the first solution time for which the force returns to zero. This should be time 1.71010E-02, with corresponding displacement 9.33837E+01 (mm).

Exiting the AUI: Choose File→Exit to exit the AUI. You can discard all changes.

Problem 52: Crushing of a crash tube

Deformed mesh:



Force-deflection curve:

