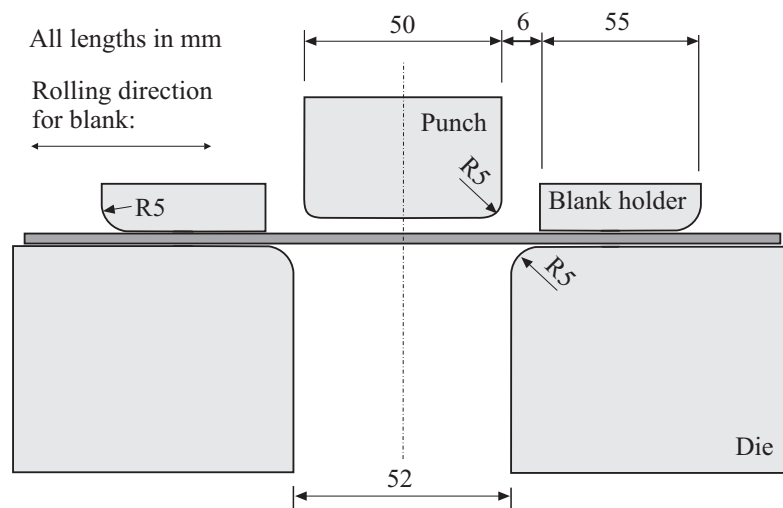


### Problem description

The figure shows a metal blank being formed by a punch, blank holder and die.



This problem is the same as the problem B3. 2-D Draw Bending, described in the following reference:

Makinouchi, Nakamachi, Onate, Wagoner, *Proceedings of the 2<sup>nd</sup> International Conference NUMISHEET '93*, 1993.

The objective of the problem is to determine the shape of the formed blank, including springback.

We analyze the case in which the blank holding force is 19.6 kN, and the blank material is high tensile strength steel with a thickness of 0.74 mm. During the forming, we move the punch at a constant punch speed of 100 mm/sec. The punch stroke is 70 mm.

We include friction with a friction coefficient of 0.129.

We will solve the problem using two separate analyses: a static-implicit analysis and a dynamic-explicit analysis.

The blank is modeled using 3D-shell elements. A large strain/large displacement formulation is used for the shell elements (ULJ formulation). The plastic-orthotropic material model is used with the following material properties:

**Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis**

---

$$E_a = E_b = E_c = 2.06 \times 10^5 \text{ N/mm}^2$$

$$\nu_{ab} = \nu_{ac} = \nu_{bc} = 0.3$$

$$G_{ab} = G_{ac} = G_{bc} = \frac{E_a}{2(1 + \nu_{ab})} = 7.923 \times 10^4 \text{ N/mm}^2$$

$$\rho = 7.8 \times 10^{-9} \text{ N-s}^2/\text{mm}^4$$

$$\sigma = 6.7716 \times 10^2 (0.01129 + \varepsilon_p)^{0.2186} \text{ N/mm}^2 \text{ (plastic stress-strain curve)}$$

$$r_0 = 1.73, r_{45} = 1.34, r_{90} = 2.24 \quad \text{(Lankford coefficients)}$$

The material *a* direction is the material rolling direction.

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

- Using the 3D-shell elements
- Using the rigid-target contact algorithm
- Using displacement loads with force unloading
- Using displacement loads with arrival time
- Using explicit time integration

**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 files `prob33_1.in` and `prob33_1.plo`. You need to copy files `prob33_1.in` and `prob33_1.plo` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

**Invoking the AUI and choosing the finite element program**

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

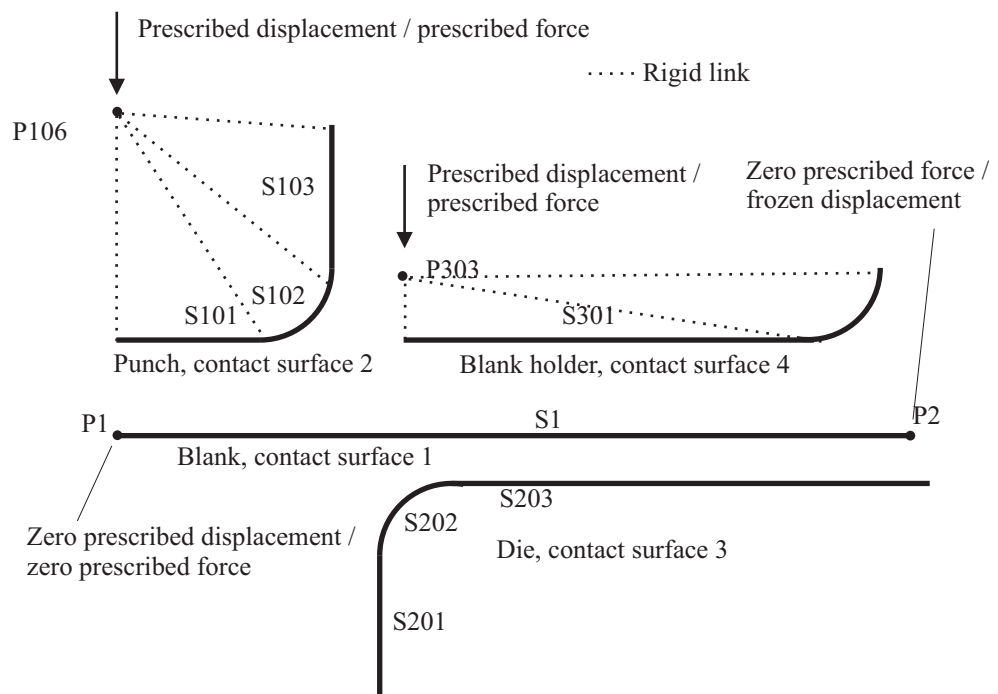
### Static-implicit analysis

First we will solve the model using a static-implicit analysis. Notice that the Analysis Type is set to Statics, so by default, the analysis is a static analysis. Static analyses are always implicit.

#### Overview of model definition

The following figure shows the key geometry used in defining this model. In this sketch, we are viewing the model from the side, so that surfaces are viewed as lines.

Only  $\frac{1}{4}$  of the actual problem is modeled, using symmetry boundary conditions.

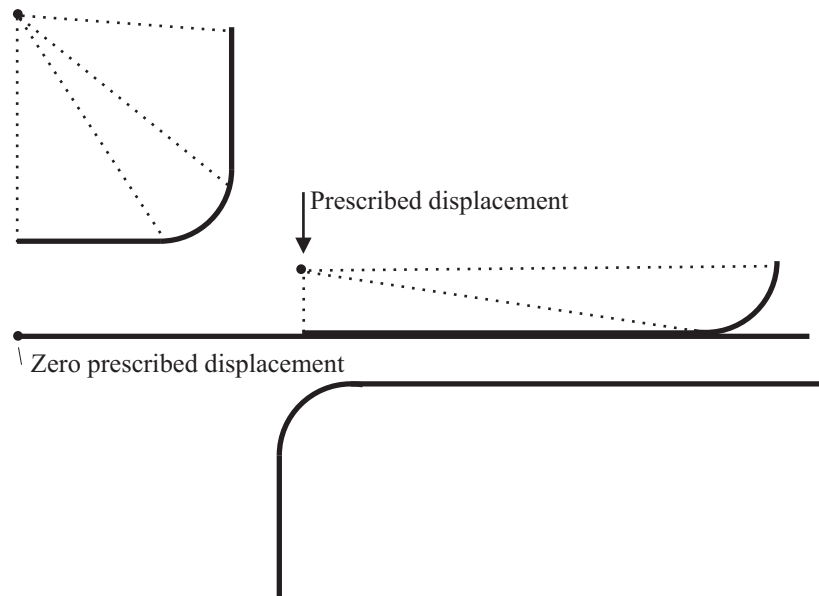


The model is solved in 10 operations, each operation taking one or more solution steps:

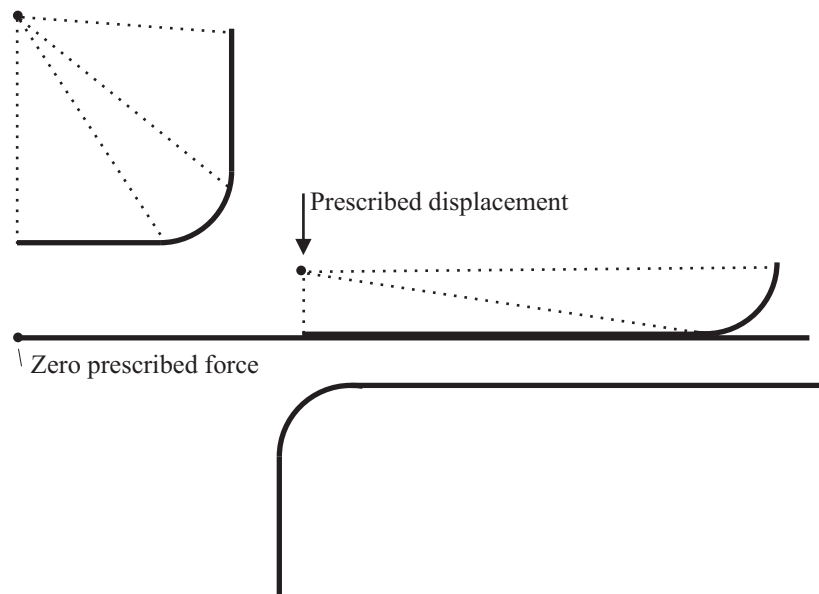
*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

- 1) Move blank holder until contact is established between blank and blank holder (1 solution step, to time 1.0).



- 2) Switch prescribed displacement on blank to zero prescribed force (1 solution step, to time 2.0).

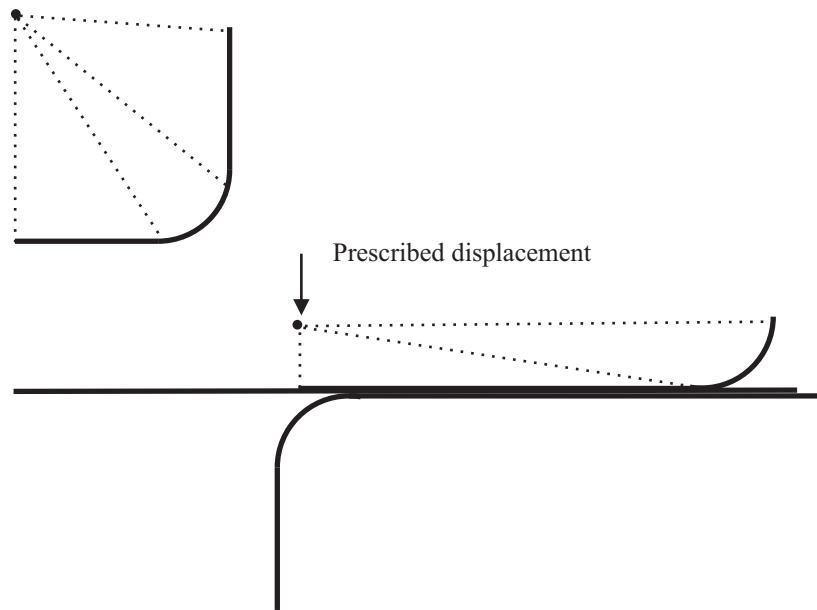


---

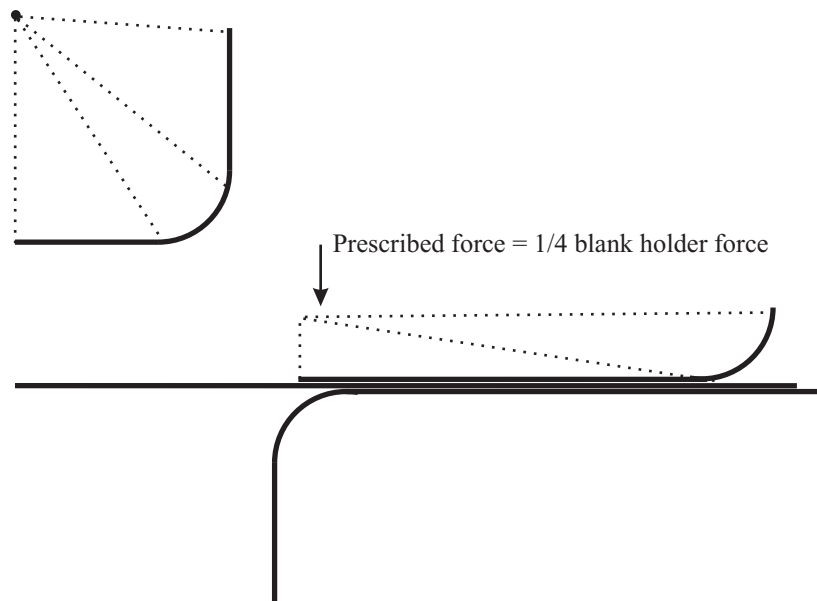
*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

- 3) Move blank holder until contact is established between blank and die (1 solution step, to time 3.0).



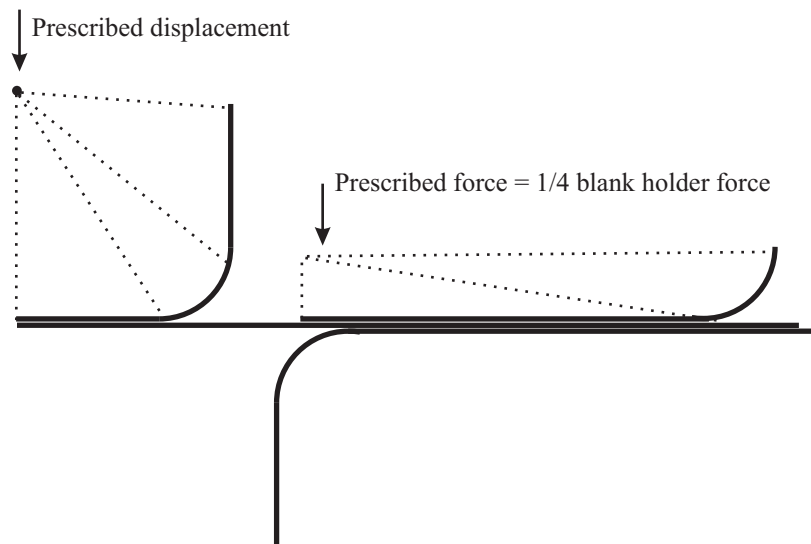
- 4) Switch to prescribed force on blank holder (1 solution step, to time 4.0).



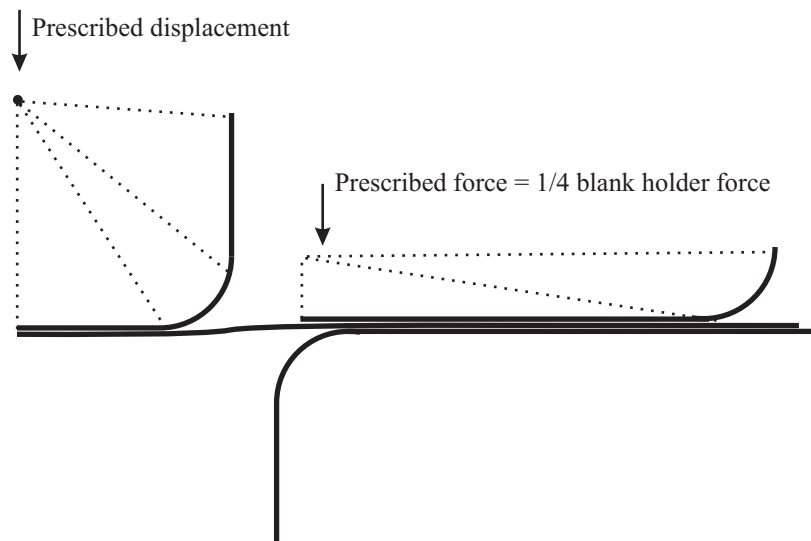
*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

- 5) Move punch to level of blank (1 solution step, to time 4.0126). In this operation, the punch speed is 100 mm/sec, the motion of the punch is 1.26 mm (downwards), so the time step size is 0.0126 sec. The punch does not contact the blank at the end of this operation.



- 6) Move punch until contact is established between blank and punch (2 solution steps, to time 4.0146). In this operation, the punch speed is 100 mm/sec, the motion of the punch is 0.2 mm, so the time step size is 0.001 sec.

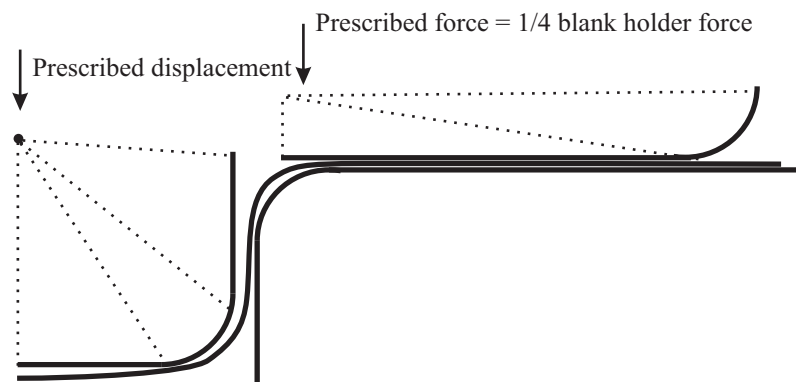


---

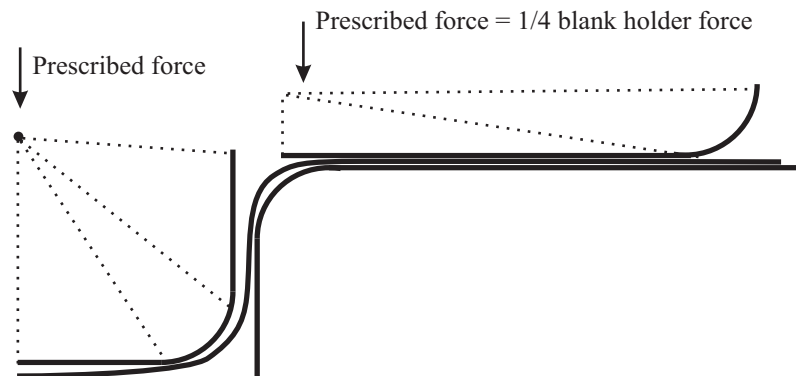
*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

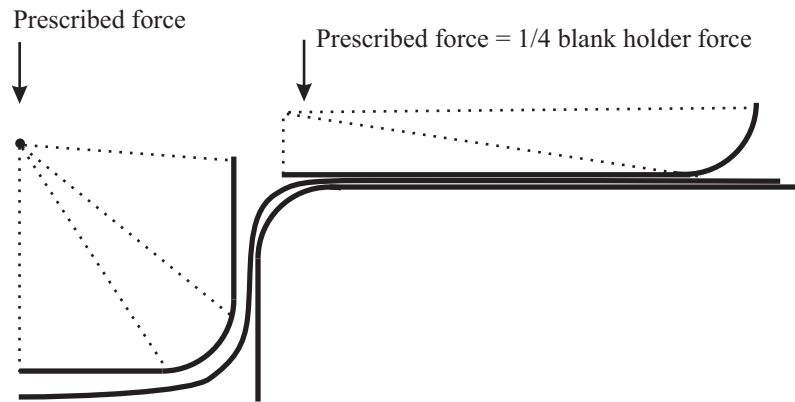
- 7) Move punch to full stroke (200 solution steps, to time 4.7126). In this operation, the punch speed is 100 mm/sec, the motion of the punch is 69.8 mm (downwards), so 199 steps with time step size 0.0035 sec and 1 step with time step size 0.0015 sec are used. The total motion of the punch in operations 5 – 7 is 71.26 mm.



- 8) Switch punch loading from prescribed displacement to prescribed force (1 solution step, step size 0.2874, to time 5.0).



9) Gradually reduce punch prescribed force (10 solution steps, to time 15.0).



10) Freeze displacement at point on blank, remove prescribed force on blank holder, remove contact group (1 solution step, to time 16.0). The “frozen” displacement is equal to the displacement at the previous solution time (time 15.0).



The most difficult operations are operations 1, 3 and 6. In each of these operations, contact is established between the blank and blank holder, die and punch, respectively. Many equilibrium iterations are required for each of these operations, and ATS cutbacks do not help in obtaining the solution. So the maximum number of iterations is set to a large number. After operation 6, contact is fully established, so fewer equilibrium iterations are required and ATS cutbacks can help in obtaining the solution. So we reduce the maximum number of iterations after operation 6, using a restart analysis.

**Defining model control data, geometry, subdivision data, boundary conditions, rigid links, displacement loads, materials and element groups**

We have prepared a batch file (prob33\_1.in) that performs the following operations:

- ▶ Sets the heading.
- ▶ Selects the automatic-time-stepping method.
- ▶ Chooses a large displacement/large strain analysis.
- ▶ Activates line searches and sets the maximum number of iterations to 999.



---

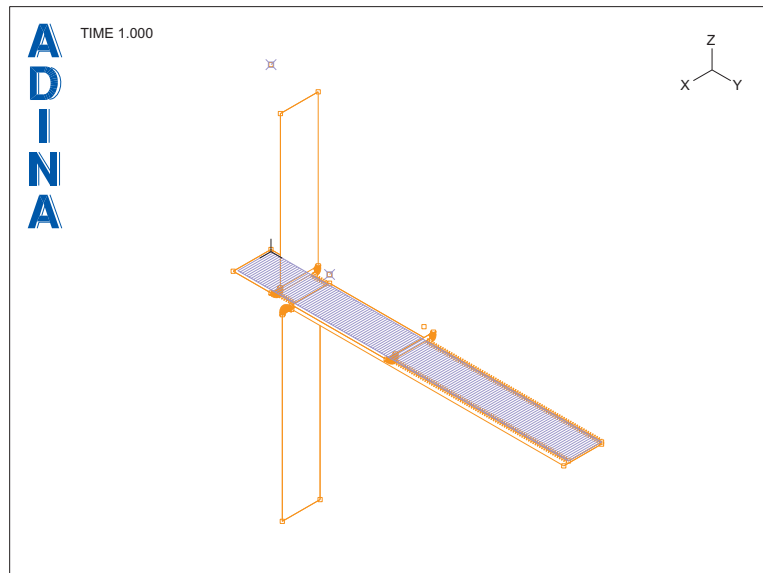
*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

- ▶ Sets iteration tolerances.
- ▶ Saves updated shell element director vectors.
- ▶ Defines fixities.
- ▶ Defines points, lines, surfaces and rigid links for the blank, punch, die and blank holder.
- ▶ Defines boundary conditions.
- ▶ Subdivides the surfaces.
- ▶ Defines the material.
- ▶ Defines a shell element group. 3D-shell elements are requested, and 3 point trapezoidal rule integration in the shell t direction is requested.
- ▶ Defines a spring element group and two soft spring elements. Each spring is placed at the master node of the rigid links, so that the AUI does not delete the degrees of freedom of the master nodes of the rigid links.

Choose File→Open Batch, navigate to the working directory or folder, select the file prob33\_1.in and click Open. The AUI processes the commands in the batch file.


The graphics window should look something like this:

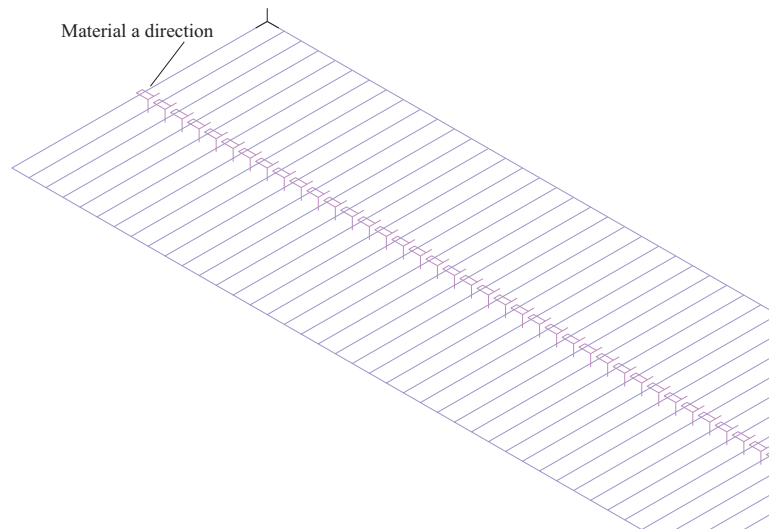


---

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*


---


We will check the material  $a$  direction (rolling direction). Press the F8 key, uncheck the Display field for the spring element group and click OK. Now click the Show Material Axes icon . When you zoom into the mesh plot, the graphics window should look something like this:



The rectangle in the material axis triad shows the material  $a$  direction.

### **Defining contact conditions**

*Contact group:* Click the Contact Groups icon , add group 1, set the Type to 3-D Contact and set the Algorithm to Rigid Target. Set the “Default Coulomb Friction Coefficient” to 0.129 and the “Contact Surface Offset” to “Shell Thickness”. Click the Advanced tab and set the Death Time to 15.5. Click the Rigid Target Algorithm tab, set the “Normal Contact Stiffness” to 1E4 and the “Maximum Sliding Velocity for Sticking Contact” to 1E-3. Click OK to close the dialog box.

*Contact surface for blank:* Click the Define Contact Surfaces icon  and add contact surface 1. In the first row of the table, set the Surf/Face # to 1, then click Save. (Do not close the Define Contact Surface dialog box yet.)

*Contact surface for punch:* Add contact surface 2, and, in the first three rows of the table, set the Surf/Face # to 101, 102, 103 respectively, then click Save. (Do not close the Define Contact Surface dialog box yet.)


---

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---



*Contact surface for die:* Add contact surface 3, then check the “Specify Orientation” button. In the first three rows of the table, set the Surf/Face # to 201, 202, 203, and also set the Orientation to “Opposite to Geometry” for all three rows. Click Save (do not close the Define Contact Surface dialog box yet.)

*Contact surface for blank holder:* Add contact surface 4, and, in the first two rows of the table, set the Surf/Face # to 301, 302, then click OK.

*Meshing the contact surfaces for the punch, die and blank holder:* Click the Mesh Rigid Contact Surface icon , set the Contact Surface to 2 and click Apply. Repeat for contact surfaces 3 and 4, then click OK.

*Defining contact pairs:* Click the Define Contact Pairs icon , then define the following contact pairs and click OK:

Contact Pair Number	Target Surface	Contactor Surface
1	2	1
2	3	1
3	4	1

When you click the Clear icon  and the Mesh Plot icon , the graphics window should look something like the figure on the next page.

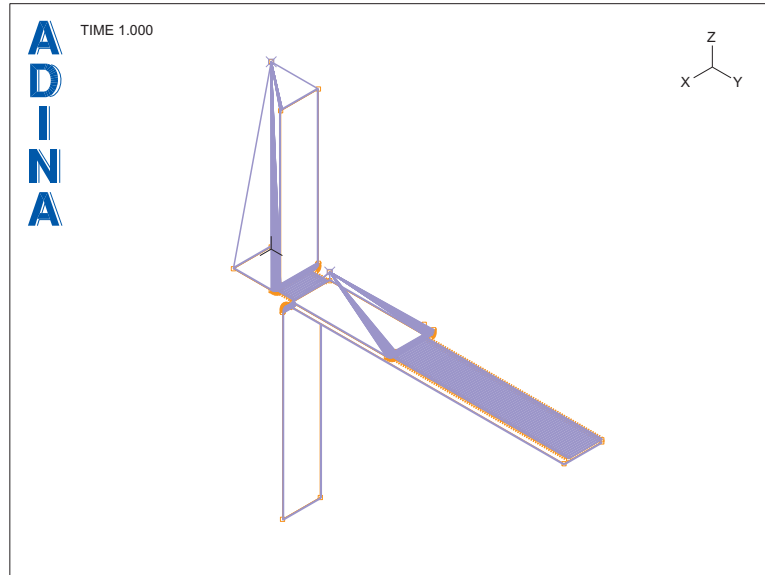
### **Defining the loads**


#### *Blank stabilizer*

The blank stabilizer displacement/force uses two time functions: time function 1 controls the prescribed displacement, time function 2 controls the prescribed force. Choose Control→ Time Function, verify that the time function 1 has constant unit value, then add time function 2, define the time function as follows and click OK.

Time	Value
0	0
16	0

**Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis**



Now click the Apply Load icon , set the Load Type to Displacement and click the Define... button. In the Define Displacement dialog box, add displacement 1, set the Z Prescribed Value of Translation to 0.0 and click OK. In the Apply Load dialog box, fill in the table as follows and click OK.

Point #	Relative To	Time Function	Arrival Time	Unloading Type	Unloading Time	Unloading Force	Unloading Time Function
1	Original (leave blank)	1	0.0 (leave blank)	Time	0.5	0.0	2
2	Deformed	1	15.5	(leave blank)			

The load on point 1 is a prescribed displacement during the solution at time 1.0. After the solution at time 1.0 is obtained, the load on point 1 switches to a prescribed force (because time 1.0 is later than the unloading time of 0.5). The magnitude of the prescribed force is 0 because time function 2 is equal to 0.

The load on point 2 is inactive until the solution at time 16.0 (because time 16.0 is the first solution time later than the arrival time of 15.5). For the solution at time 16.0, the load on point 2 is a prescribed displacement with value 0 (because the z prescribed value of translation is 0). The prescribed displacement is measured relative to the deformations of the model at time 15.0. The effect is to freeze the motion of point 2 at time 16.0.

---

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

*Blank holder force*


The blank holder displacement/force uses two time functions: time function 3 controls the prescribed displacement, time function 4 controls the prescribed force. Choose Control→ Time Function, add time function 3, define it as

Time	Value
0	0
1	0.65
2	0.65
3	1.27
16	1.27

and click Save. Then add time function 4, define it as

Time	Value
0	1.0
15	1.0
16	0.0

and click OK.

Now click the Apply Load icon , set the Load Type to Displacement and click the Define... button. In the Define Displacement dialog box, add displacement 2, set the Z Prescribed Value of Translation to -1.0 and click OK. In the Apply Load dialog box, set the Load Number to 2, fill in row 1 of the table as follows and click OK.

Point #	Relative To	Time Function	Arrival Time	Unloading Type	Unloading Time	Unloading Force	Unloading Time Function
303	Original (leave blank)	3	0.0 (leave blank)	Time	2.5	-4900	4

The load on point 303 is a prescribed displacement controlled by time function 3 up to and including solution time 3.0. After the solution at time 3.0 is obtained, the load on point 303 switches to a prescribed force (because time 3.0 is later than the unloading time of 2.5). The magnitude of the prescribed force is 4900 multiplied by time function 4, so this prescribed force remains 4900 until solution time 15, and then becomes zero at solution time 16.

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

*Punch displacement*


The punch displacement/force uses two time functions: time function 5 controls the prescribed displacement, time function 6 controls the prescribed force. Choose Control→Time Function, add time function 5, define it as

Time	Value
0	0
1	0
4	0
4.7126	71.26
16	71.26

and click Save. Then add time function 6, define it as

Time	Value
0	0
4.7126	0
5	1
15	0.1
16	0

and click OK.

Now click the Apply Load icon  (a green square with red dashed arrows), set the Load Type to Displacement and set the Load Number to 2, fill in row 2 of the table as follows and click OK.

Point #	Relative To	Time Function	Arrival Time	Unloading Type	Unloading Time	Unloading Force	Unloading Time Function
106	Original (leave blank)	5	0.0 (leave blank)	Time	4.9	0	6



The load on point 106 is a prescribed displacement controlled by time function 5 up to and including solution time 5.0. After the solution at time 5.0 is obtained, the load on point 106 switches to a prescribed force (because time 5.0 is later than the unloading time of 4.9). The magnitude of the prescribed force is equal to the reaction force at this point at solution time 5.0, multiplied by time function 6.

### Defining the time steps

In this run, we will solve operations 1 to 6, up to solution time 4.0146. Choose Control→Time Step, fill in the table as follows and click OK.

Number of steps	Magnitude
4	1
1	0.0126
2	0.001

### Generating the data file, running ADINA Structures

Click the Save icon  and save the database to file prob33. Click the Data File/Solution icon , set the file name to prob33\_ima, make sure that the Run Solution button is checked and click Save.

The ADINA Structures run finishes in 7 steps.

When ADINA Structures is finished, close all open dialog boxes.



### Restart analysis

We will continue the analysis using the restart feature.


Choose Control→Solution Process, set the "Analysis Mode" to "Restart Run" and click OK. Choose Control→Solution Process, click the Method button, set the Maximum Number of Iterations to 100 and click OK twice to close both dialog boxes. Now choose Control→Time Step, edit the table to read as follows, then click OK.

Number of steps	Magnitude
199	0.0035
1	0.0015
1	0.2874
10	1
1	1



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

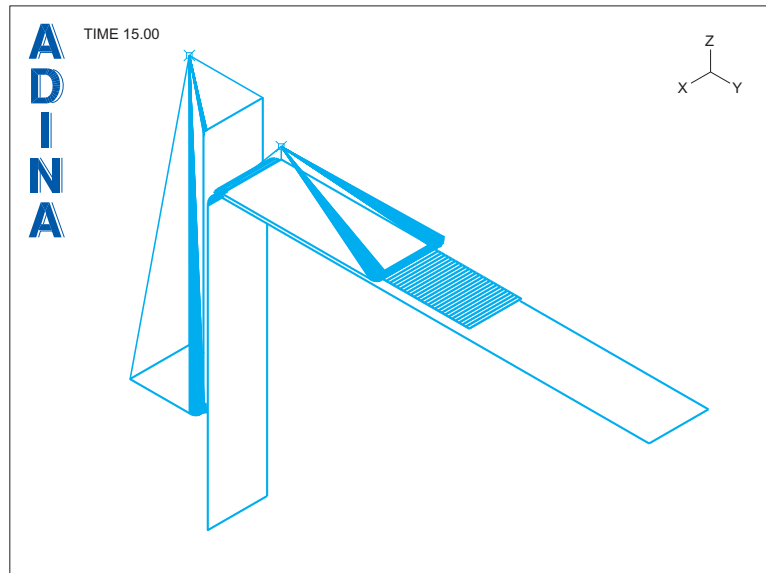
Click the Save icon  to save the database. Click the Data File/Solution icon , set the file name to prob33\_imb, make sure that the Run Solution button is checked and click Save. The AUI opens a window in which you specify the restart file from the first analysis. Enter restart file prob33\_ima and click Copy.

The ADINA Structures run finishes in 212 steps (step 219 is the last solution step).

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 prob33\_imb.

The graphics window shows the solution for the last time step. In this step, the contact group is dead, and the blank holder force pushes the rigid links of the blank holder downwards.

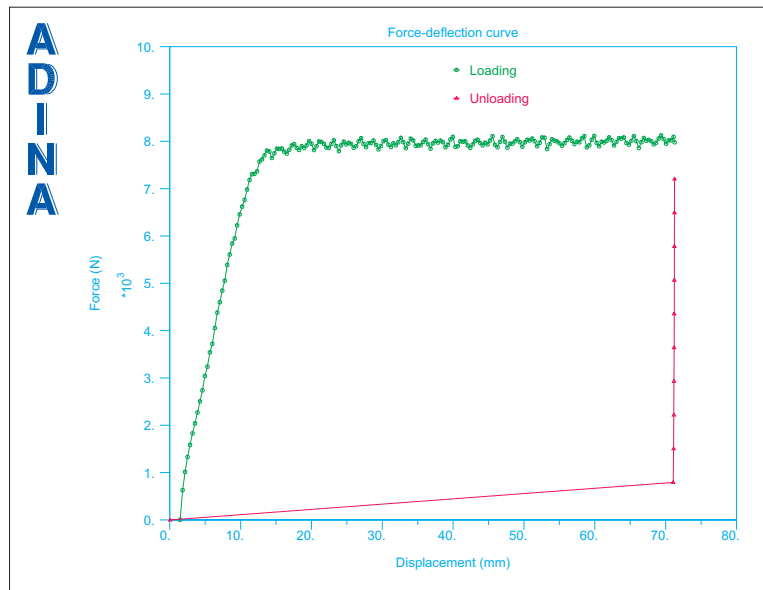
Click the Previous Solution icon  once and click the Refit icon . The graphics window should look something like this:






### Post-processing

*Force-deflection curve:* We have put all of the post-processing instructions into file prob33\_1.p1o. Choose File→Open Batch, navigate to the working directory or folder, select the file prob33\_1.p1o and click Open. The AUI processes the commands in the batch file. The graphics window should look something like this:

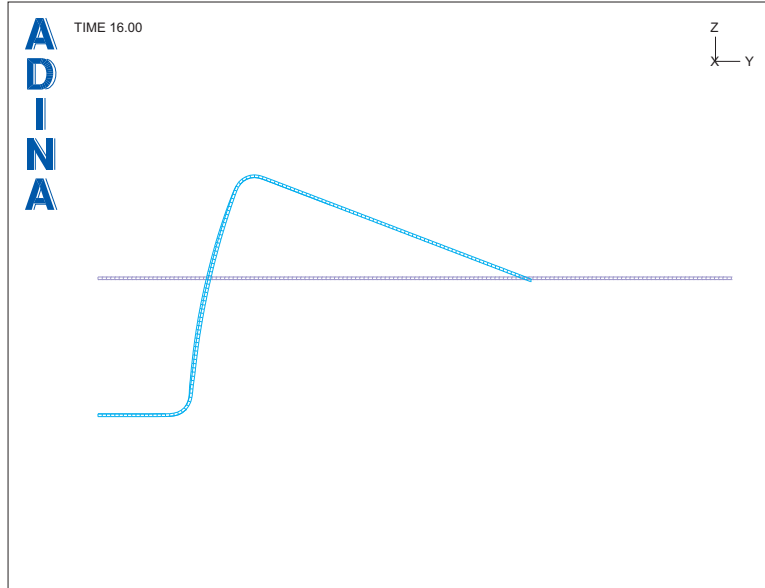


The graphics window shows the force-deflection curve. In this plot, we have multiplied the force by 4 to account for symmetry. The green curve shows the loading, and the loading is determined from the reaction force on the punch displacement. The red curve shows the unloading, and the unloading is determined from the prescribed punch force.

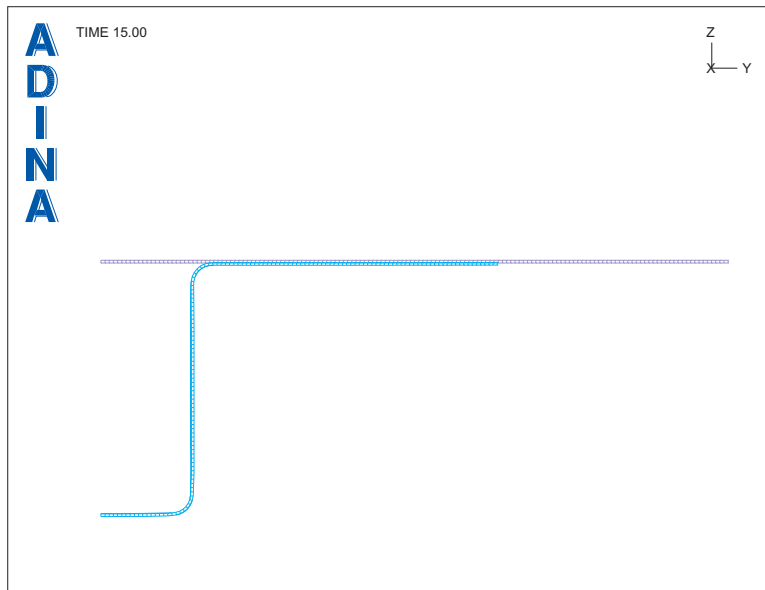
*Final blank shape:* Click the Batch Continue icon . The graphics window should look something like the top figure on the next page.

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---



The graphics window shows the side view of the blank, after springback. Click the Previous Solution icon ◀ to show the blank before springback. The graphics window should look something like this:




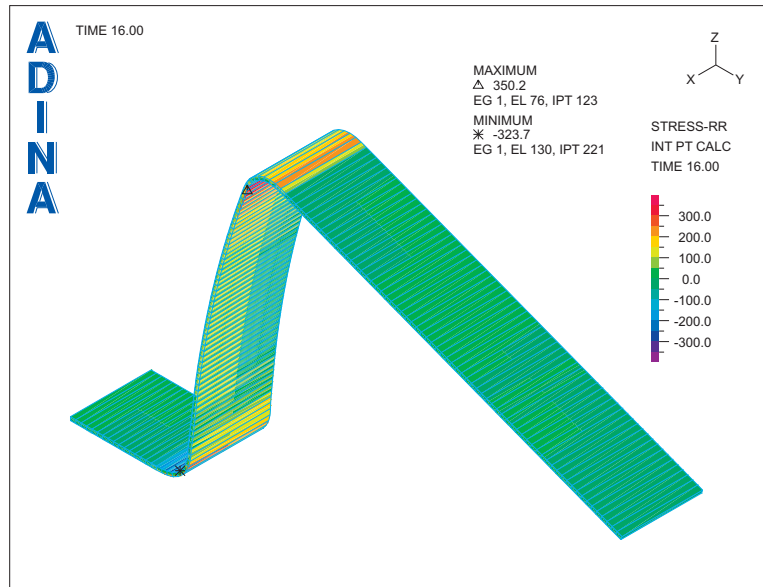
---


Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis

---

During the springback process, the node on the right of the blank does not move vertically. This is because we froze the displacement of this node during springback.

*Residual stresses:* Click the Batch Continue icon . The graphics window should look something like this:

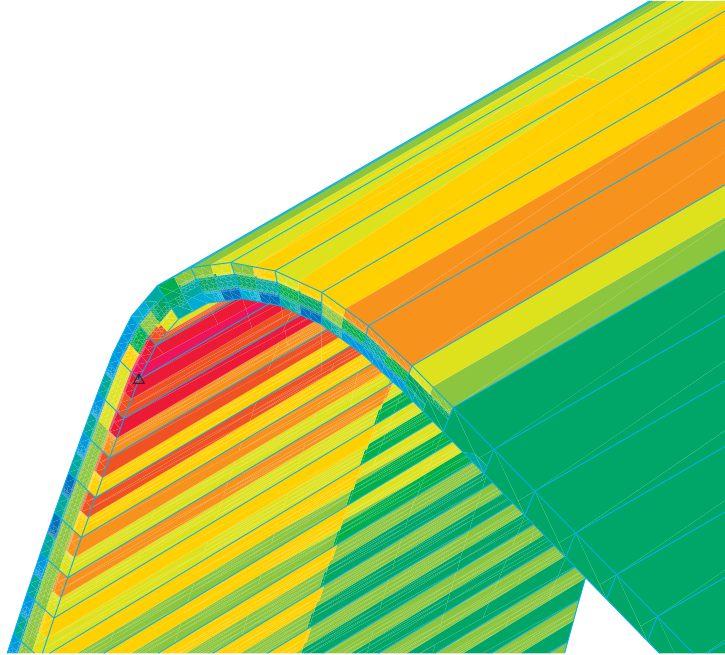



The graphics window shows the residual stresses in the rolling direction. The blank is plotted with the true shell thicknesses. The residual stresses are plotted without interpolation from the integration points. Use the Zoom icon  to enlarge the blank near the top bend. The graphics window should look something like the figure on the next page.

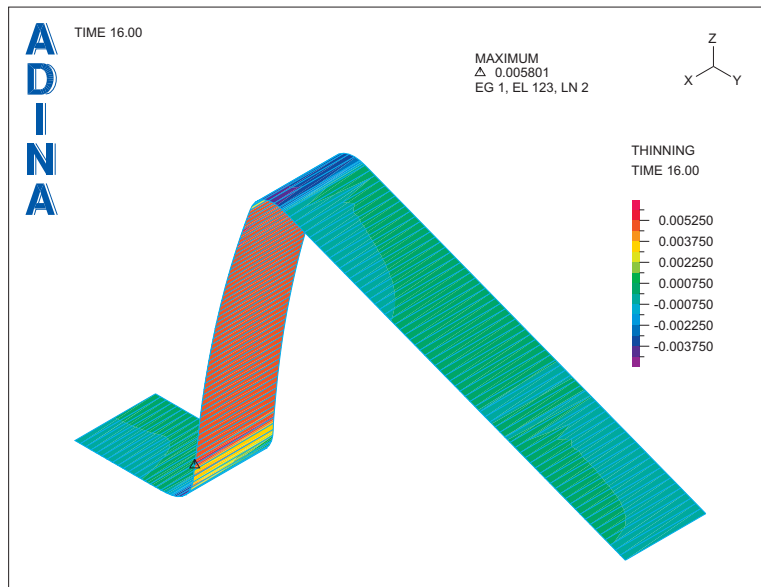
You can clearly see that the residual stress at each integration point through the thickness is different (3 integration points through the thickness are used).

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---



*Thinning:* Click the Batch Continue icon . The graphics window should look something like this. The maximum thinning is about 0.6%.



## Dynamic-explicit analysis

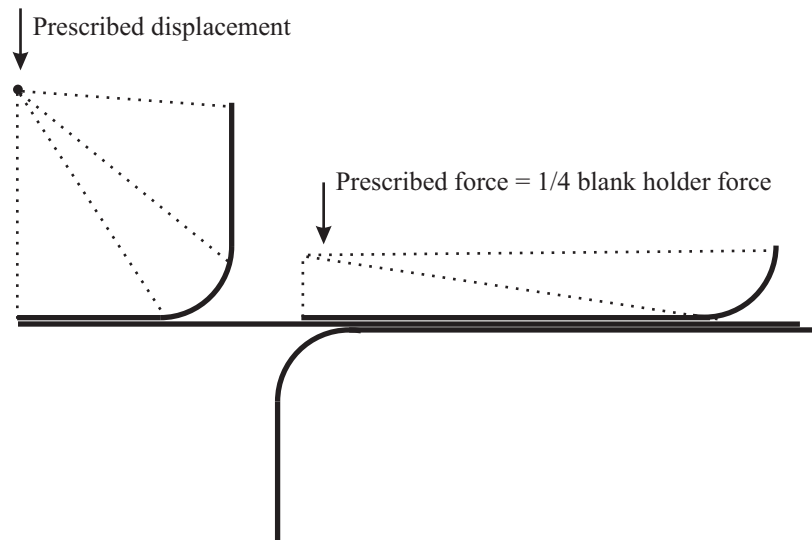
Now let's solve the model using an explicit analysis.

We present this solution for illustrative purposes only. In order to keep the number of explicit time steps reasonable, we use a very large mass scaling factor. This causes the solution to be inaccurate, as compared to the static solution.

### Overview of model definition

The model is solved in 9 operations, each operation taking one or more solution steps:

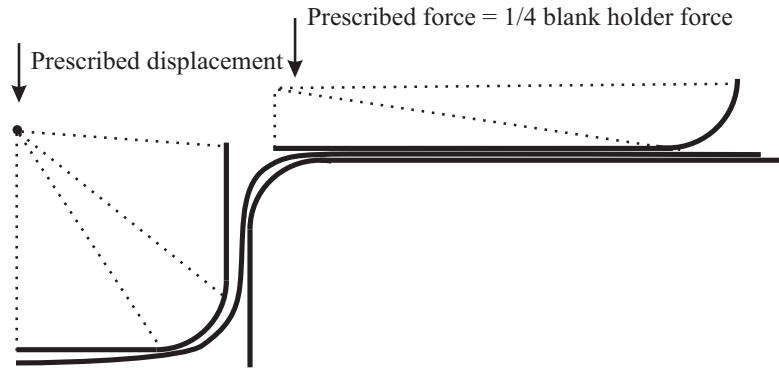
1) to 4) Static analysis. Same as for the static-implicit analysis (to solution time 4.0126). These operations provide a solution that is an initial condition for the explicit analysis.



5) Restart to explicit analysis. Move punch to full stroke (200 steps requested, requested step size 0.0035, to solution time 4.7126). In this operation, the punch speed is 100 mm/sec. ADINA Structures automatically calculates the time step actually used. In order to increase the critical time step size, and therefore reduce the number of time steps required, we use a mass scaling factor of  $10^5$ .

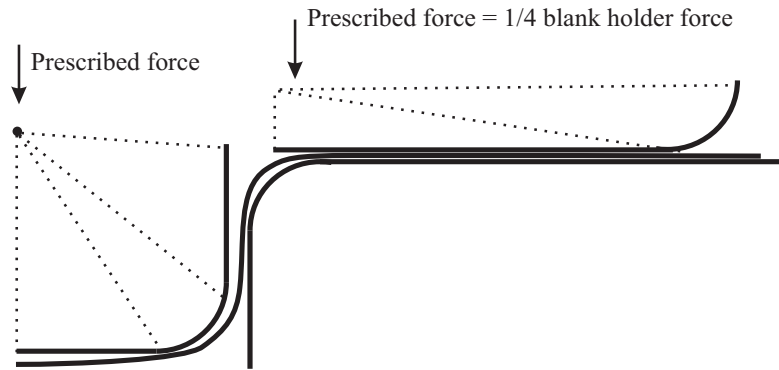
*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

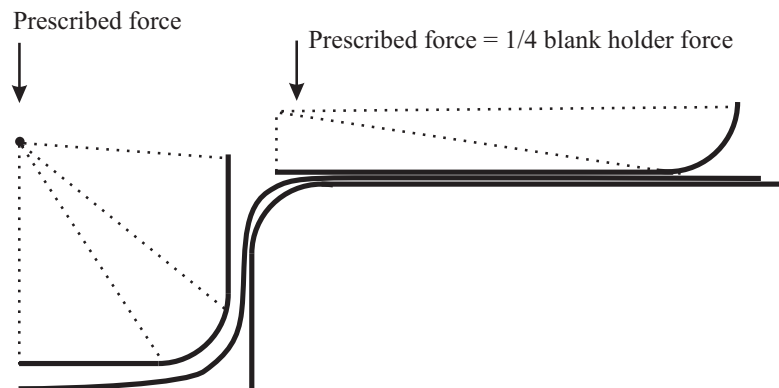


6) Restart to dynamic-implicit analysis (1 solution step, step size 0.0035, to time 4.7161).

7) Switch punch loading from prescribed displacement to prescribed force (1 solution step, step size 0.2839, to time 5.0).



8) Gradually reduce punch prescribed force (10 solution steps, to time 15.0).



---

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

9) Freeze displacement at point on blank, remove prescribed force on blank holder, remove contact group (1 solution step, to time 16.0). The “frozen” displacement is equal to the displacement at the previous solution time (time 15.0).





In operations 7 to 9, large dynamic time steps are used, to simulate static conditions. Hence, at the end of operation 9, we obtain the “static” springback shape.

Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose file prob33.ldb from the recent file list near the bottom of the File menu.

**Static-implicit analysis to obtain initial conditions for the explicit analysis**

Choose Control→Solution Process, set the "Analysis Mode" to "New Run" and click OK. Choose Control→Solution Process, click the Method button, set the Maximum Number of Iterations to 999 and click OK twice to close both dialog boxes. Now choose Control→Time Step, edit the table to read as follows, then click OK.

Number of steps	Magnitude
4	1
1	0.0126


Click the Save icon  to save the database. Click the Data File/Solution icon , set the file name to prob33\_exa, make sure that the Run Solution button is checked and click Save.

The ADINA Structures run finishes in 5 steps.

When ADINA Structures is finished, close all open dialog boxes.

**Restart to explicit analysis**

Choose Control→Solution Process, set the "Analysis Mode" to "Restart Run" and click OK.

Set the Analysis Type to Dynamics-Explicit and click the Analysis Options icon . Set the Global Mass Scaling Factor to 1E5 and click OK to close the dialog box.



---

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

Now choose Control→Time Step, edit the table to read as follows, then click OK.

Number of steps	Magnitude
200	0.0035



Click the Save icon  and save the database to file prob33. Click the Data File/Solution icon , set the file name to prob33\_exb, make sure that the Run Solution button is checked and click Save. The AUI opens a window in which you specify the restart file from the first analysis. Enter restart file prob33\_exa and click Copy.

ADINA Structures automatically computes the time step size. About 13000 time steps are required to solve this model. When ADINA Structures is finished, close all open dialog boxes.

### Restart to dynamic-implicit analysis

Set the Analysis Type to Dynamics-Implicit. Now choose Control→Time Step, edit the table to read as follows, then click OK.

Number of steps	Magnitude
1	0.0035
1	0.2839
10	1
1	1

Click the Save icon  to save the database. Click the Data File/Solution icon , set the file name to prob33\_exc, make sure that the Run Solution button is checked and click Save. The AUI opens a window in which you specify the restart file from the second analysis. Enter restart file prob33\_exb and click Copy.

The ADINA Structures run finishes in about 13 steps (solution step 13280).

### Post-processing

When ADINA Structures is finished, close all open dialog boxes and choose Post-Processing from the Program Module drop-down list (you can discard all changes). Now choose File→Open Porthole, select file prob33\_exa.por, set “Load” to “Partial Sequence of Files Starting with Specified File”, set “Number of Files to Load in Sequence” to 3 and click Open.


We will post-process this model using the same file as we used for the static analysis. Choose File→Open Batch, navigate to the working directory or folder, select the file prob33\_1.plo



---

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

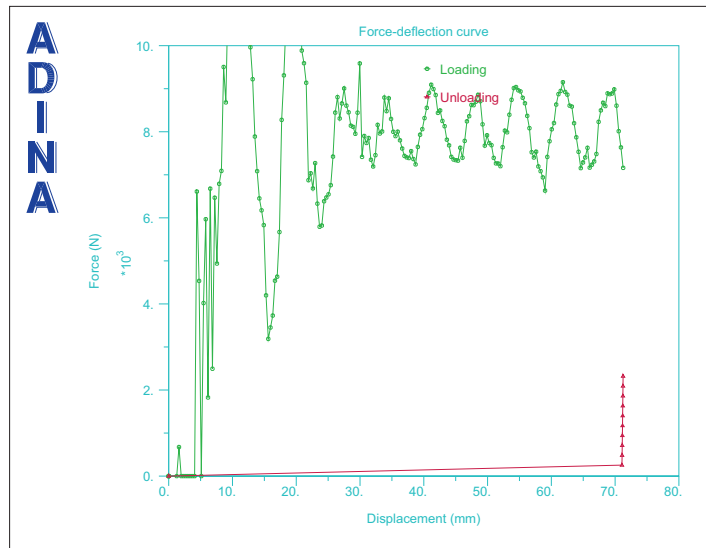
---

and click Open. The AUI processes the commands in the batch file. Click the Batch Continue icon  to show all of the plots. The plots are shown in the figures on the next two pages.

*Exiting the AUI:* Choose File→Exit (you can discard all changes).

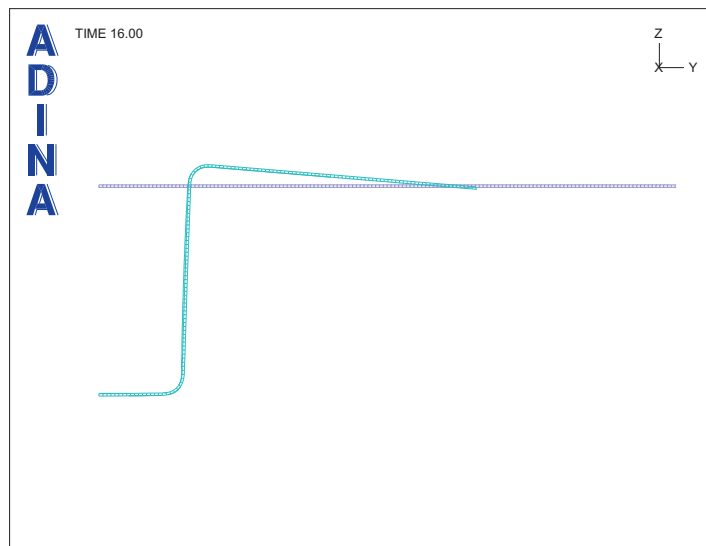
**Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis**

*Force-deflection curve:*



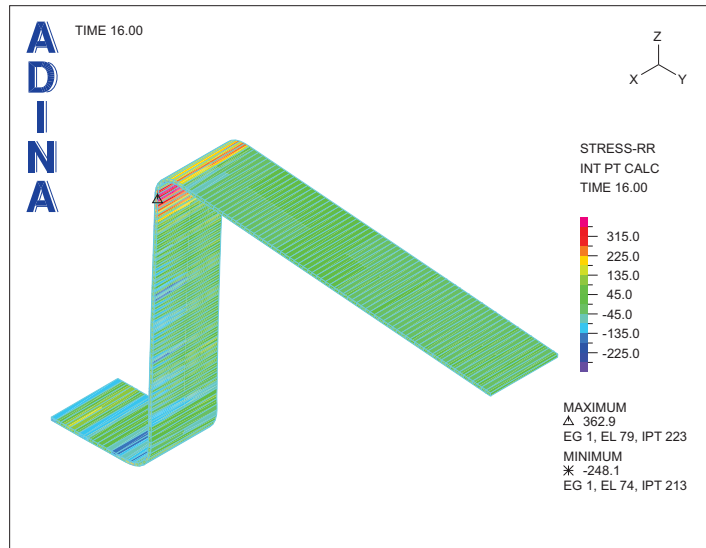
Notice that the loading force oscillates around the same steady-state value as in the static analysis.

*Final blank shape:*

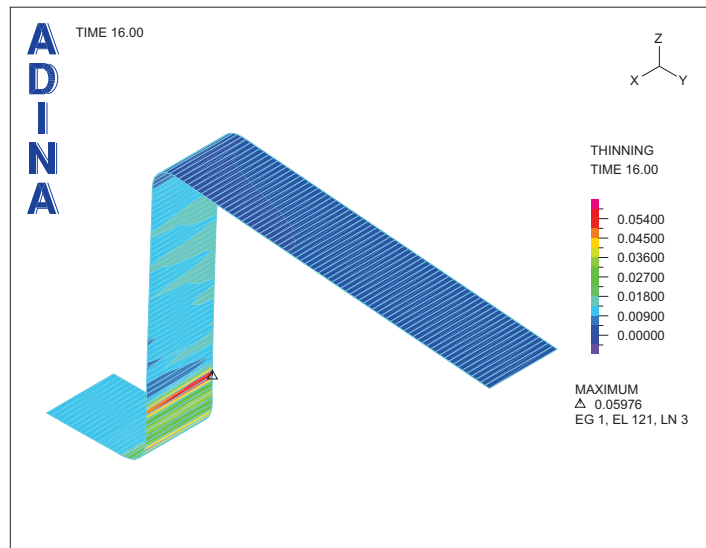


Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis

Residual stresses:



Thinning:



There are thin bands of relatively large thinning. These bands form early in the explicit analysis, probably because the inertia forces are very large.

*Problem 33: U-bending of a metal sheet – static-implicit and dynamic-explicit analysis*

---

This page intentionally left blank.