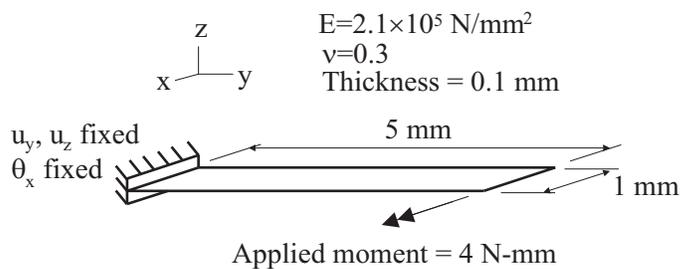


### Problem description

An initially flat plate is bent by an applied moment:



The moment is large enough to require a large displacement analysis.

The purpose of this problem is to demonstrate some of the consequences of using 6 degree of freedom nodes in large displacement shell elements. You might want to reread Section 2.7.3 of the ADINA Structures Theory and Modeling Guide to review the use of 5 degree of freedom and 6 degree of freedom nodes in shell analysis.

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

- Specification of the shell nodes drilling stiffness
- Use of beam elements to control the shell nodes drilling stiffness

### 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 `prob45_1.in` and `prob45_2.in`. You need to copy files `prob45_1.in` and `prob45_2.in` 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.

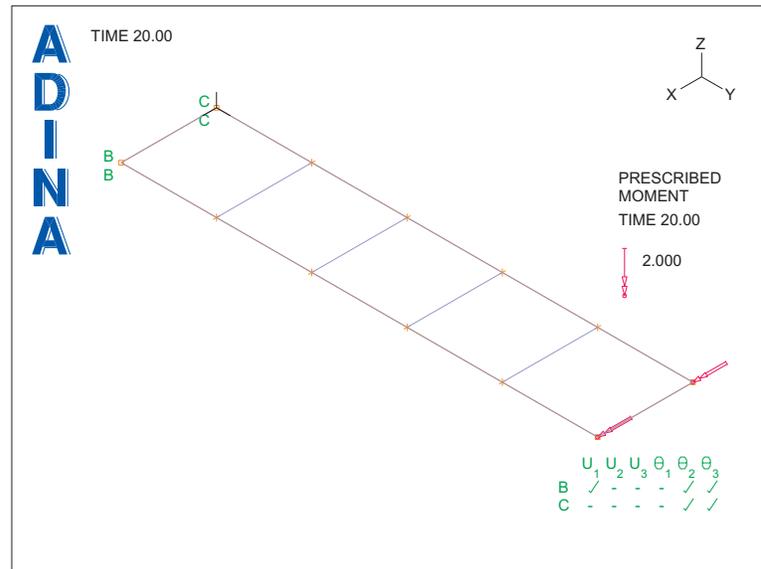
*Problem 45: Large displacement bending of a plate by an applied moment*

---

**Model definition**

We have prepared a batch file (prob45\_1.in) which contains all of the model definition. Shell elements are used to model the plate. The moment is applied in 20 time steps.

Choose File→Open Batch, navigate to the working directory or folder, select the file prob45\_1.in and click Open. The graphics window should look something like this:



Notice that the y and z displacement, and the x rotation, are fixed on the clamped end. In addition, the x displacement is fixed at one point on the clamped end to remove the rigid-body translation in the x direction.

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

Click the Save icon  and save the database to file prob45. Click the Data File/Solution icon , set the file name to prob45, make sure that the Run Solution button is checked and click Save.

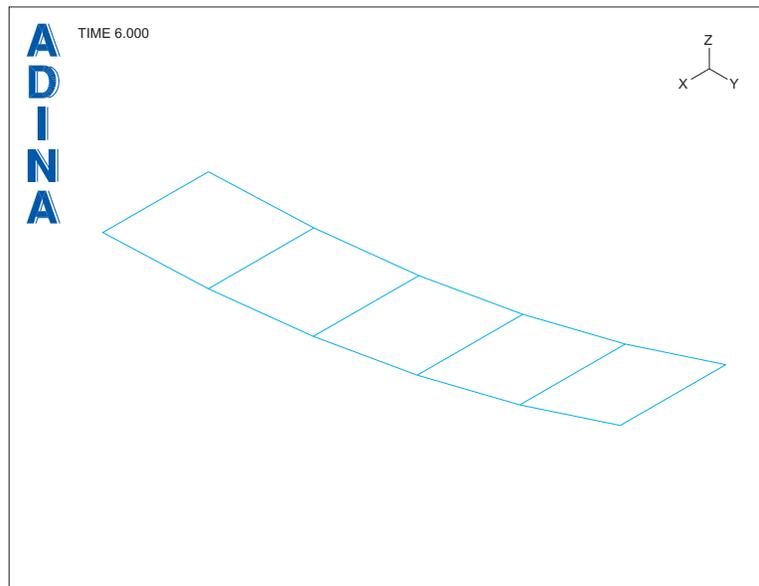
The model does not converge in the 7<sup>th</sup> time step.

---

*Problem 45: Large displacement bending of a plate by an applied moment*

---

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 prob45. The graphics window should look something like this:



The model deformations look fine. So the plot does not give us any clues about why the model did not converge.

### **Examining the ADINA Structures output file**

To figure out what happened, we need to examine the ADINA Structures output file. If you didn't close the ADINA Structures window, click the View Output button. If you already closed the ADINA Structures window, then use a text editor, such as `vi` or Notepad, to open file `prob45.out`.

Don't close the AUI.

In the output file, look for the text

```
STEP NUMBER = 7
```

and examine the "NORM OF INCREMENTAL ROTN." column for the equilibrium iterations of step 7. These values should be similar to those shown in the following table.

*Problem 45: Large displacement bending of a plate by an applied moment*

---

Iteration	NORM OF INCREMENTAL ROTN	NODE-DOF	MAX VALUE
0	4.26E-01	1-Z	2.03E-01
1	3.01E-01	12-Z	1.83E-01
2	2.99E-01	11-Z	-1.77E-01
3	2.88E-01	11-Z	-1.74E-01

From this table, we see that the norm (square root of the sum of the squares) of all of the incremental rotations in the model for iteration 0 is 4.26E-01 and the maximum incremental rotation in iteration 0 is 2.03E-01 at node 1. Since the rotations are measured in radians, the incremental rotations at nodes 1, 11 and 12 appear to be very large.

Evidently the nodes at the clamped end and the nodes with the applied moment are assigned 6 degrees of freedom. At all of the nodes with 6 degrees of freedom, there is nothing in the model to provide stiffness to the drilling degrees of freedom (the drilling degrees of freedom are the rotations in the direction of the shell normals). So we need to provide stiffness for those degrees of freedom.

In this model, the AUI assigns 6 degrees of freedom to the nodes on the clamped end because there are both free rotations (the y- and z-rotations) and fixed rotations (the x rotations) on the clamped end. The AUI assigns 6 degrees of freedom to the nodes with applied moment, because of the applied moment.

**Pre-processing: clamped nodes**

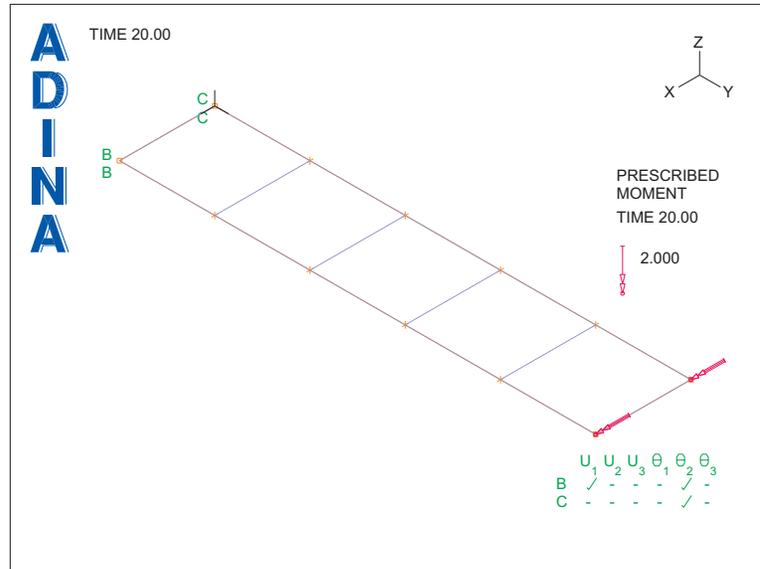
For the clamped nodes, there is no reason to leave the z rotation free. Let's see what happens if we fix the z rotation for the clamped nodes.

*Invoking the pre-processor:* Set the Program Module drop-down list to ADINA Structures (you can discard all changes). Choose prob45. idb from the recent file list near the bottom of the File menu.

Click the Apply Fixity icon  and notice that fixities P1 and P2 are applied to geometry points 1 and 2. Click the Define... button, set the Fixity Name to P1, check the Z-Rotation button, set the Fixity Name to P2, check the Z-Rotation button and click OK twice to close both dialog boxes.

When you click the Redraw icon , the graphics window should look something like the figure on the next page.

*Problem 45: Large displacement bending of a plate by an applied moment*



Notice that the Z rotation is fixed at the nodes on the clamped end.

**Generating the ADINA Structures data file, running ADINA Structures, examining the output file**

Click the Save icon . Click the Data File/Solution icon , set the file name to prob45, make sure that the Run Solution button is checked and click Save. You can replace the existing prob45.dat file.

Again, the model does not converge in the 7<sup>th</sup> time step. We need to look at the ADINA Structures output file again. Click the View Output button in the ADINA Structures window, look for the text

STEP NUMBER = 7

and examine the “NORM OF INCREMENTAL ROTN.” column. These values should be similar to those shown in the following table.

Iteration	NORM OF INCREMENTAL ROTN	NODE-DOF	MAX VALUE
0	3.11E-01	11-Z	-1.92E-01
1	2.81E-01	11-Z	-1.77E-01
2	3.09E-01	12-Z	2.08E-01
3	6.01E-01	12-Z	-3.85E-01

*Problem 45: Large displacement bending of a plate by an applied moment*

---

Nodes 11 and 12 appear to have large incremental rotations.

At nodes 11 and 12, we cannot fix the z rotations because the drilling stiffness direction will change as the model deforms, and the drilling stiffness direction will no longer coincide with the z direction.

**Pre-processing: drilling stiffness**

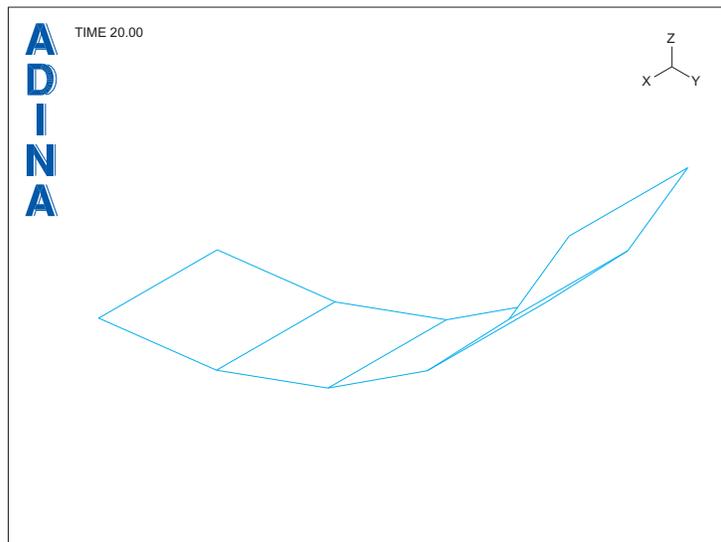
*Returning to the pre-processor:* Close all open windows and dialog boxes.

By default, ADINA Structures assigns a small amount of drilling stiffness to nodes without drilling stiffness. Let's increase the amount of drilling stiffness. Choose Control→Miscellaneous Options, and, in the Shell Options box, set the Stiffness Factor to 0.001 and click OK.

**Generating the ADINA Structures data file, running ADINA Structures, examining the output file**

Click the Save icon . Click the Data File/Solution icon , set the file name to prob45, make sure that the Run Solution button is checked and click Save. You can replace the existing prob45.dat file.

This time, ADINA Structures runs for 20 steps. 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 prob45. The graphics window should look something like this:



### Post-processing

*Rotations and moment reactions:* Choose List→Value List→Zone, set Variable 1 to (Displacement: X-ROTATION), Variable 2 to (Reaction: X-MOMENT\_REACTION) and click Apply. Scroll down to display the results at time 20. The X moment reaction at node 1 is -1.99760 and the X moment reaction at node 2 is -1.99761. The X rotation at nodes 11 and 12 is 1.19038 (radians). Click OK to close the dialog box.

The moment reactions nearly balance the applied moment of 4.0. A comparison analytical solution is

$$\theta = \frac{ML}{EI} = 1.14 \text{ (radians)}$$

For many purposes, the solution is “good enough”. But we can obtain a better solution by introducing a soft beam element to eliminate the drilling stiffness.

### Pre-processing: soft beam element

*Invoking the pre-processor:* Set the Program Module drop-down list to ADINA Structures (you can discard all changes). Choose prob45.odb from the recent file list near the bottom of the File menu.

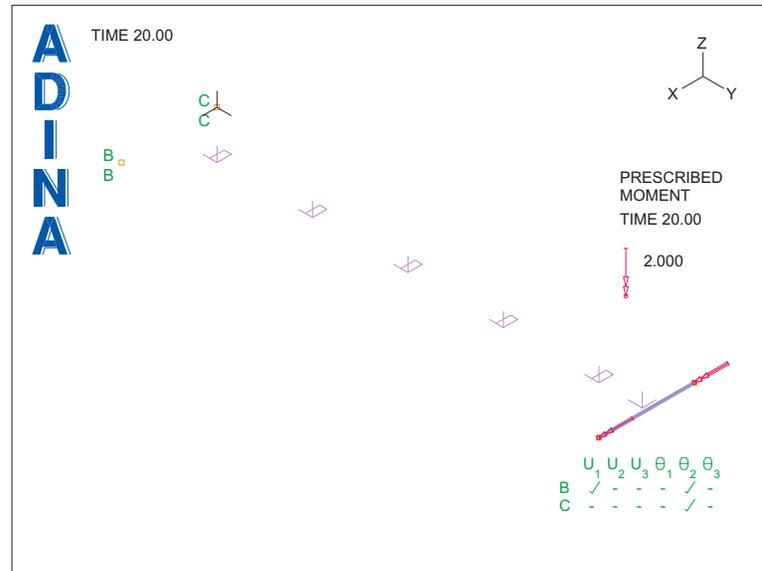
We have prepared a batch file (prob45\_2.in) which contains the beam element definition. Choose File→Open Batch, navigate to the working directory or folder, select the file prob45\_2.in and click Open. We can't see the beam element because it lies on one of the shell element lines. Click the No Mesh Lines icon  to show just the beam element. Click the Modify Mesh Plot icon , click the Element Depiction... button, click the “Display Local System Triad” button and click OK twice to close both dialog boxes. The graphics window should look something like the figure on the next page.

The beam element triad shows that the beam element s direction lies in the plane of the shells, and the beam element t direction is perpendicular to the plane of the shells.

Click the Cross Sections icon . Notice that the only non-zero area moment of inertia is the tt moment of inertia, and that the cross-sectional area is very small. So the beam only has bending stiffness for a rotational direction perpendicular to the plane of the shells (in other words, for rotations in the shell normal direction), and the beam has no axial stiffness.

*Problem 45: Large displacement bending of a plate by an applied moment*

---



Since the beam now provides stiffness in the drilling stiffness direction, we don't need any shell drilling stiffness. Choose Control→Miscellaneous Options, uncheck the "Assign Stiffness to Nodes with Zero Stiffness" button and click OK.

**Generating the ADINA Structures data file, running ADINA Structures, post-processing**

Click the Save icon . Click the Data File/Solution icon , set the file name to prob45, make sure that the Run Solution button is checked and click Save. You can replace the existing prob45.dat file.

Again, ADINA Structures runs for 20 steps. 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 prob45. Choose List→Value List→Zone, set Variable 1 to (Displacement: X-ROTATION), Variable 2 to (Reaction: X-MOMENT\_REACTION) and click Apply. Scroll down to display the results at time 20. The X moment reaction at nodes 1 and 2 is  $-1.99999$ , and the X rotation at nodes 11 and 12 is  $1.16531$  (radians). Click OK to close the dialog box.

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

**Notes**

1) If we had turned off the drilling stiffness feature before running the original model, the solution would have stopped immediately with a "zero pivot" message.

---

*Problem 45: Large displacement bending of a plate by an applied moment*

---

2) It is safe to leave the y rotation of the clamped nodes free, because the y rotation direction lies in the plane of the shell, and hence has stiffness.

3) In the comparison analytical solution, the plate constant  $D = \frac{Et^3}{12(1-\nu^2)}$  is not used,

because the y rotations are free and the plate can undergo anticlastic bending. The plate responds as if it were a beam.

4) The drilling stiffness feature affects the solution slightly, because a small amount of the applied moment acts in the direction of the drilling stiffness (this is because the anticlastic effect bends the shell normals so that the normals have components in the x direction). Because the drilling stiffness is implemented using grounded rotational spring elements, a small amount of the applied moment is taken by the drilling stiffness springs.

On the other hand, when the beam element is used, all of the applied moment goes into the finite element model, so the reactions at the clamped nodes are in equilibrium with the applied moment.

*Problem 45: Large displacement bending of a plate by an applied moment*

---

This page intentionally left blank.