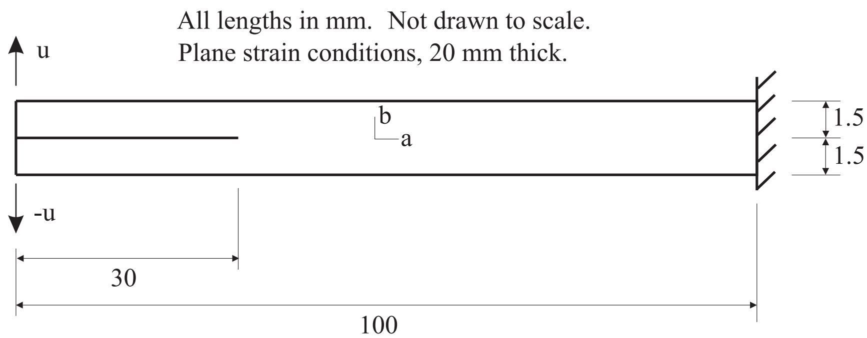
Problem description

The figure shows a double cantilever beam (DCB) of a composite material, subjected to displacement loads at its ends.



Material properties: E_{aa}=135300 N/mm^2
E_{bb}=E_{cc}=9000 N/mm^2
G_{ab}=5200 N/mm^2
ν_{ab}=ν_{ac}=0.24
ν_{bc}=0.46

Interface properties: G_{IC}=0.28 N/mm
t_n=57 N/mm^2

We are interested in the force-deflection curve, considering delamination of the material.

This problem can be found in the following reference:

> G. Alfano and M. A. Crisfield, "Finite element interface models for the delamination analysis of laminated composites: mechanical and computational issues", *Int. J. Num. Meth. Engng,* 2001; 50; 1701-1736.

This problem involves only mode I delamination, thus only mode I interface properties are required.

The Poisson's ratios require some discussion. In orthotropic analysis, there are two possible conventions used for the Poisson's ratios. These are

1) $e_b = -\dfrac{\nu_{ba}}{E_a}\sigma_a, \qquad \dfrac{\nu_{ba}}{E_a} = \dfrac{\nu_{ab}}{E_b}$

2) $e_b = -\dfrac{\nu_{ab}}{E_a}\sigma_a, \qquad \dfrac{\nu_{ba}}{E_b} = \dfrac{\nu_{ab}}{E_a}$

In these formulas, e_b is the strain in direction *b* due to a uniaxial stress σ_a. ADINA Structures uses convention 1). The reference does not state which convention is used, but if convention 1) is assumed, then the constitutive matrix is not positive-definite. Therefore we suppose that convention 2) is used in the reference. When we transform the Poisson's ratios from convention 2) to convention 1), we obtain $\nu_{ab} = \nu_{ac} = 0.016$, $\nu_{bc} = 0.46$, and these values will be used in the analysis.

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

- Defining an orthotropic material
- Defining cohesive interfaces
- Specifying low-speed dynamics
- Setting the Rayleigh damping factors for element groups

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 cannot be solved with the 900 nodes version of the ADINA System because there are too many nodes in the model.

Much of the input for this problem is stored in the following files: `prob57_1.in`, `prob57_1.plo`. You need to copy these files 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.

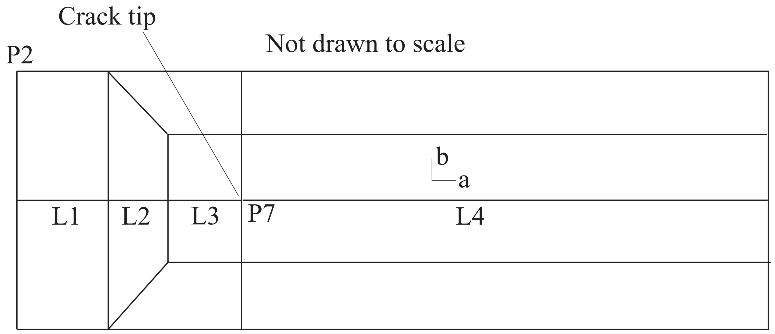
Model definition

We have prepared a batch file (`prob57_1.in`) that defines the following items:

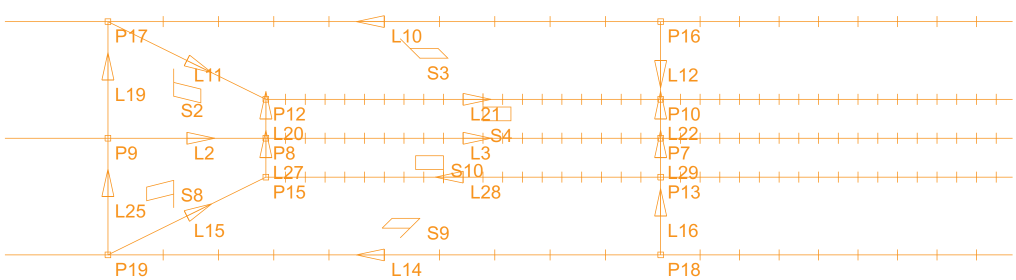
► Problem heading
► Control data, including solution tolerances. This is a large displacement analysis. The automatic time-stepping method is used. However, low-speed dynamics is not used (we will turn on the low-speed dynamics option later.)
► Geometry points, lines, surfaces
► Subdivisions of the lines and surfaces
► Boundary conditions
► Loads. Prescribed displacements are used to model displacement-controlled loading.
► 100 time steps of size 0.01. The full displacement of 8 mm is applied at time 1.0.

► Element group 1, which is a plane strain element group. Note that in ADINA Structures, only unit thickness is considered in plane strain analysis. Therefore the reaction forces corresponding to the prescribed displacements need to be multiplied by 20 to obtain the force-deflection curve for the original problem.

For future reference, we give the geometry point, line and surface numbers that will be used below:



Choose File→Open Batch, navigate to the working directory or folder, select the file `prob57_1.in` and click Open. Use the Zoom icon and enlarge the region near the crack tip (geometry point 7). The graphics window should look something like this:



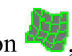
Material properties

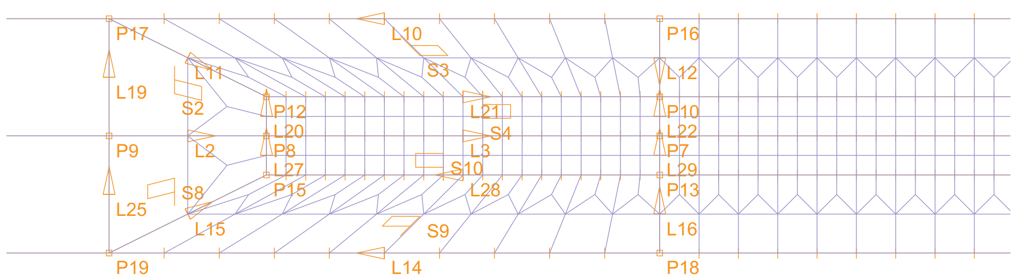
Click the Manage Materials icon and click the Elastic Orthotropic button. Add material 1, and, in the Young's Modulus box, set a to 135300, b to 9000 and c to 9000. In the Poisson Ratio box, set ab to 0.016, ac to 0.016 and bc to 0.46. In the Shear Modulus box, set ab to 5200, then click OK. The AUI displays a warning message concerning GAC and GBC. Click OK to close the warning message, then click Close to close the Manage Material Definitions dialog box.

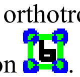
We also need to define the directions of the orthotropic axes. Choose Model→Orthotropic Axes Systems→Define, add System 1, set the "Vector Aligned with Local X-Axis" to (0, 1, 0), the "Vector Lying in the Local XY-Plane" to (0, 0, 1) and click OK. Now choose Model→Orthotropic Axes Systems→Assign (Material), edit the table so that surfaces 1 to 12 are assigned axes-system 1 and click OK.

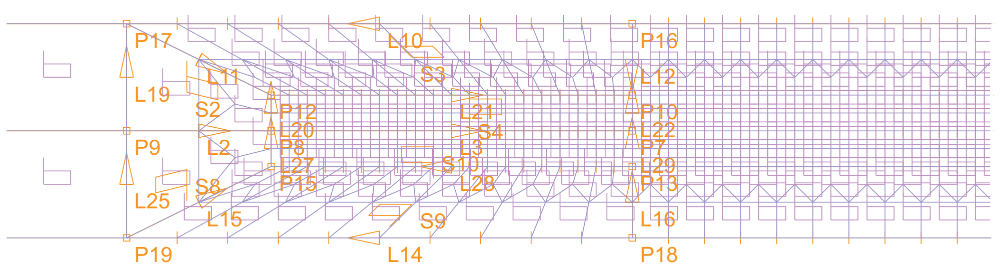
Nodes and elements

We would like to create node 1 at geometry point 2. (This will make it easier to post-process the model since we can then plot the results for node 1.) Choose Meshing→Create Mesh → Point, enter 2 in the first row of the table and click OK.

Now click the Mesh Surfaces icon , set the Nodes per Element to 4, enter 1 to 12 in the first 12 rows of the table (you might want to use the Auto... button) and click OK. The graphics window should look something like this:



Let's make sure that the orthotropic axes in the elements are correctly defined. Click the Show Material Axes icon . The graphics window should look something like this:



Notice that all of the axes point in the same direction. Click the Show Material Axes icon  again to hide the material axes.

Split definition

Up to this point, the mesh is compatible along lines 1, 2, 3, 4. In other words, there is only one node at each station along these lines. We need to split the mesh along lines 1, 2, 3, so that there are duplicate nodes at each station.

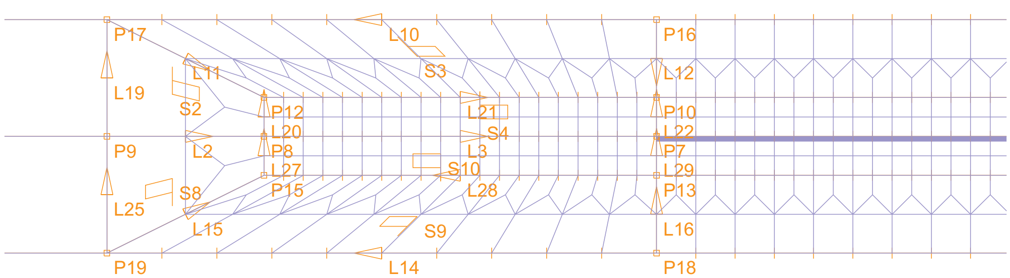
Choose Model→Cohesive Interface→Split Interface and add Split Interface Number 1. Enter 1, 2, 3 in the first three rows in the table and click OK.

Cohesive interface definition

We need to define a cohesive interface along geometry line 4. Choose Model→Cohesive Interface→Define and add Cohesive Interface Number 1. Enter 4 in the first row of the table and click OK.

Now choose Model→Cohesive Interface→Properties and add Property Set # 1. In the Fracture Toughness box, set Mode I and Mode II to 0.28, and in the Cohesive Strength of Interface box, set Normal and Shear to 57. Set the Penalty Stiffness to 1e6 and click OK. (Note, for this problem, it is not necessary to set any of the parameters in the Mixed-Mode Interaction box because this problem only undergoes Mode I delamination.)

When you click the Redraw icon , the graphics window should look something like this:



The cohesive interface is drawn with a thick line.

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

Click the Save icon and save the database to file `prob57`. Click the Data File/Solution icon, set the file name to `prob57`, make sure that the Run Solution button is checked and click Save.

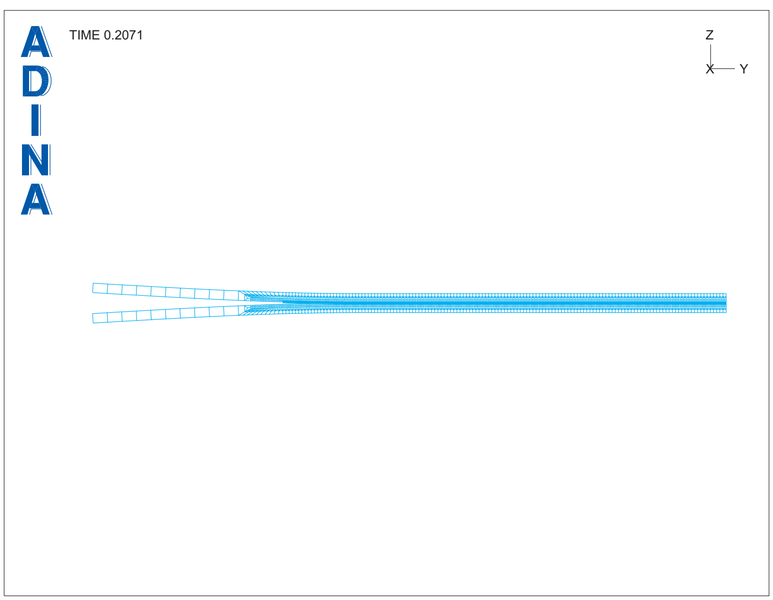
The program stops in step 21 with messages similar to

```
Mesh too distorted, Jacobian determinant not positive
Porthole file updated, nodal results saved
Porthole file updated, element results saved
*** Program stopped abnormally ***
*** Please see the *.out file for details ***
```

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 `prob57`.

Post-processing

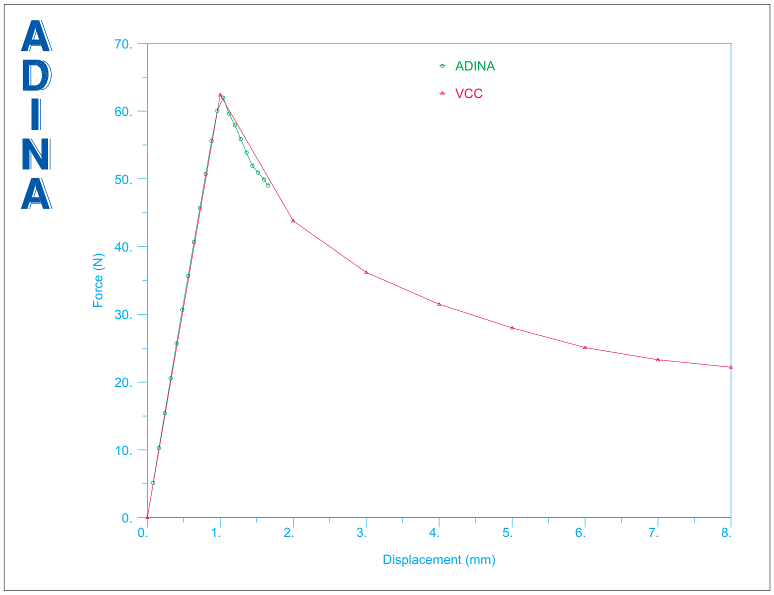
The graphics window should look something like this:



Delamination has started to occur.

Force-deflection curve

In order to obtain the force-deflection curve, we need to plot the reaction force at node 1 vs the displacement at node 1. We have put the commands for plotting the force-deflection curve in file `prob57_1.plo`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob57_1.plo` and click Open. The AUI processes the commands in the batch file. The graphics window should look something like this:



Curve VCC contains the results from Fig 12 of the reference, for the solution using the VCC method. The comparison is very good for the range of solutions obtained by ADINA Structures.

Pre-processing: low-speed dynamics

In order to obtain a solution for larger prescribed displacements, we need to activate the low-speed dynamics option of the automatic time-stepping feature.

Invoking the pre-processor: Set the Program Module drop-down list to ADINA Structures (you can discard all changes). Choose `prob57.idb` from the recent file list near the bottom of the File menu.

Click the Analysis Options icon and click the ... button to the right of the "Use Automatic Time Stepping (ATS)" field. Set "Use Low-Speed Dynamics" to "On Element Groups" and click OK twice to close both dialog boxes. Now choose Control→Analysis Assumptions→

Rayleigh Damping, and, in the first row of the table, set the Element Group to 1, "Factor, Alpha" to 0 and "Factor, Beta" to 1E-4 and click OK.

(If we had used the option "Use Low-Speed Dynamics = On Whole Model", then damping would also have been applied to the cohesive interfaces.)

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

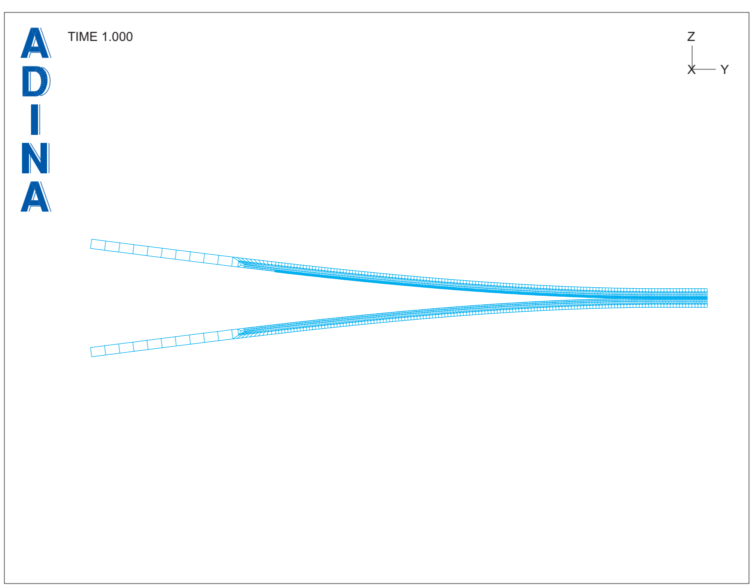
Click the Save icon. Click the Data File/Solution icon, set the file name to `prob57`, make sure that the Run Solution button is checked and click Save.

The problem now runs for all 100 steps. Each step consists of two sub-steps since the Bathe method of implicit time integration is used in low-speed dynamics.

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 `prob57`.

Post-processing

The graphics window should look something like this:

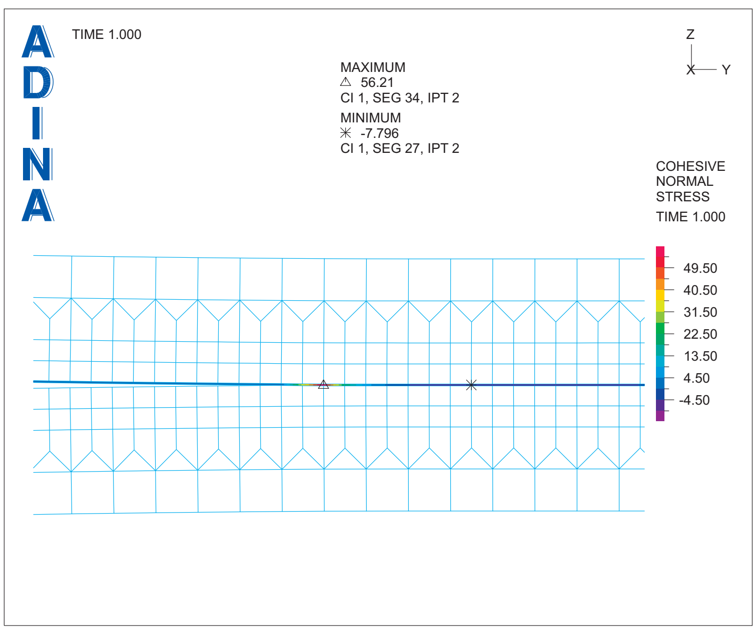


Significant delamination has occured.

Recall that we split the mesh along geometry lines 1 to 3. The plot shows that the mesh splitting is correctly defined, with duplicate nodes along lines 1 to 3. In addition, the plot shows that duplicate nodes were created along the cohesive interface line (line 4). (All of these duplicate nodes were created during generation of the .dat file.)

Cohesive normal stress

Click the Create Band Plot icon , set the Variable to (Stress: COHESIVE_NORMAL_STRESS) and click OK. Use the Mesh Zoom icon to enlarge the region of the mesh near the largest cohesive stress. The graphics window should look something like this:

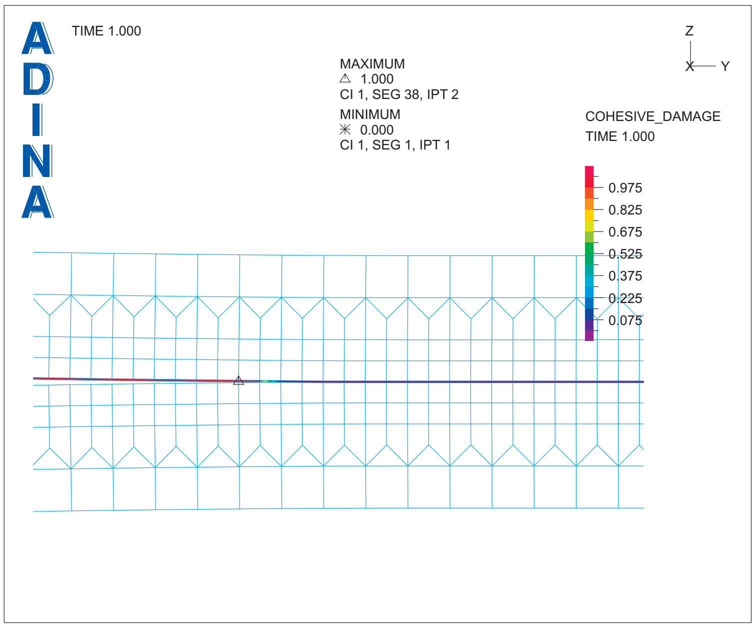


We observe that the highest normal stress occurs at the current location of the crack front. Use the icons that change the solution time to observe the crack front location for different solution times. Then click the Last Solution icon to display the last solution.

Cohesive damage

Click the Modify Band Plot icon and set the Variable to (Failure Criterion: COHESIVE_DAMAGE). Click the Band Table... button, and, in the Value Range box, set the Maximum and Minimum to Automatic, then click OK twice to close both dialog boxes.

The graphics window should look something like this:



Undamaged material has a damage value of 0, fully damaged material has a damage value of 1.

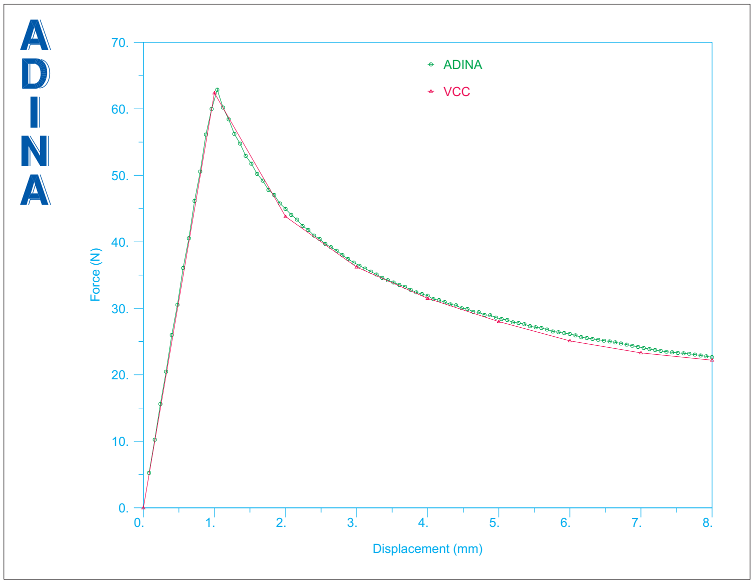
Force-deflection curve

We can plot the force-deflection curve using the same procedure used above. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob57_1.plo` and click Open. The graphics window should look something like the figure on the next page.

The comparison is very good between the ADINA Structures solution and the reference.

Choose Graph→List and look at the results for solution time 1.30000E-01. The displacement is 1.04 (mm) and the corresponding reaction force is 6.30911E+01 (N). Click Close to close the dialog box.

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


ADINA
70.
60.
50.
40.
30.
20.
10.
0.
Force (N)
0.
1.
2.
3.
4.
5.
6.
7.
8.
Displacement (mm)
ADINA
VCC

This page intentionally left blank.