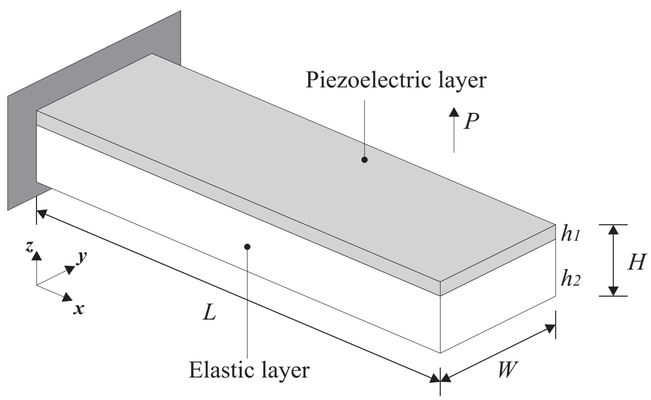
**Problem description**

The cantilever shown below is composed of two layers: the top layer is a piezoelectric material and the bottom layer is an elastic material. The polarization (P) of the piezoelectric material is along the *z*-direction. The interface between the two layers is grounded (voltage equals zero).




The dimensions of the cantilever are: $L = 0.1$ m , $W = 0.01$ m , $h_1 = 0.001$ m , $h_2 = 0.004$ m .
The material properties of the bottom layer are: $E = 90\times10^9$ Pa , $\nu = 0.3$ .
The piezoelectric material properties are:

Elastic constants: $E_x = 61\times10^9$ Pa , $E_y = 61\times10^9$ Pa , $E_z = 53.2\times10^9$ Pa ,
$\nu_{xy} = 0.35$ , $\nu_{xz} = 0.38$ , $\nu_{yz} = 0.38$ ,
$G_{xy} = 22.593\times10^9$ Pa , $G_{xz} = 21.1\times10^9$ Pa , $G_{yz} = 21.1\times10^9$ Pa
Coupling constants (N/Vm): $e_{1z} = -7.209$ , $e_{2z} = -7.209$ , $e_{3z} = 15.118$ ,
$e_{5x} = 12.332$ , $e_{6y} = 12.332$
(where 1=*x*, 2=*y*, 3=*z*, 4=*xy*, 5=*xz*, 6=*yz*)
Dielectric constants (C/Vm): $\varepsilon_{xx} = 1.53\times10^{-8}$ , $\varepsilon_{yy} = 1.53\times10^{-8}$ , $\varepsilon_{zz} = 1.5\times10^{-8}$ .

Note that the piezoelectric material properties given above shows that the polarization is along the global *z*-direction.

We will perform three different analyses:

*(1). Sensing analysis 1:* A prescribed downward displacement 0.005 m is applied at the free end and we investigate the voltage and electric field results. The material axes of the piezoelectric material coincide with the global directions.

*(2). Sensing analysis 2:* Repeat case 1 but the material axes of the piezoelectric material do not coincide with the global directions. The material properties are transformed accordingly, so that the same results as in analysis (1) are obtained.

*(3). Actuating analysis:* A uniform voltage=100 V is applied on the top surface of the piezoelectric layer and we investigate the induced displacement.

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

- Defining the 3D solid element with piezoelectric option
- Defining the piezoelectric material with different polarization direction
- Defining the electrical loading and boundary conditions
- Defining and applying an axes-system to define the material axes
- Plotting voltage and electric field results

**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 (there are too many nodes in this model).

Much of the input for this problem is stored in the following files: `prob59_1.in`, `prob59_2.in`, `prob59_3.in`. 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.

## Sensing analysis 1

**Model definition**

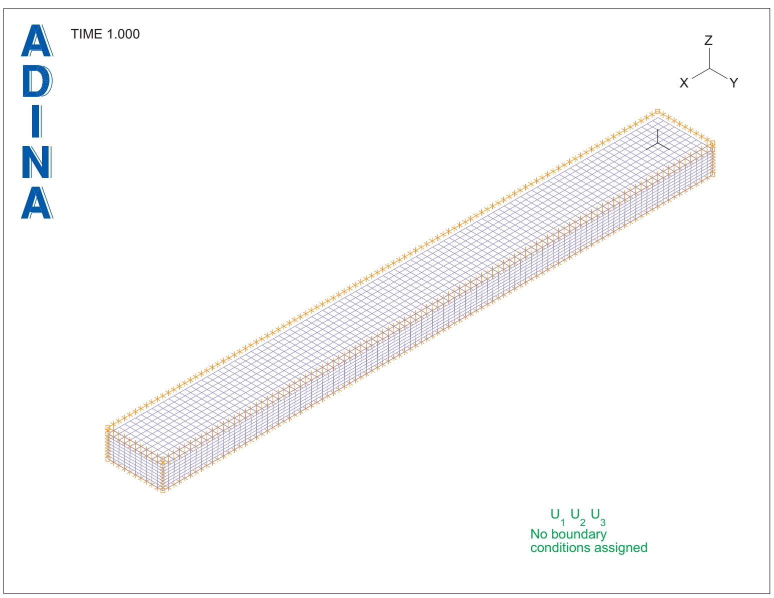
We have prepared a batch file (`prob59_1.in`) that defines the following items:

► Problem heading
► Control data, set active degrees of freedom to be X-, Y- and Z-translations. When there is a piezoelectric material, the voltage degree of freedom is automatically activated.
► Incompatible modes formulation used.
► Geometry points, lines, surfaces and volumes. Two volumes are created: the top layer is a piezoelectric material and the bottom one is an elastic material.
► Definition of the material properties for element group 1 (bottom layer), which is an

isotropic elastic material with $E$=90 GPa and $\nu$=0.3.

- ► Subdivision data for both volumes. There are 100 subdivisions along the L direction and 10 along the W direction for both volumes; and there are 8 and 2 subdivisions along the H direction, respectively, for the elastic layer and the piezoelectric layer.
- ► Definition of element group 1.
- ► Mesh generation of elements in element group 1.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob59_1.in` and click Open. The graphics window should look something like this:




**Piezoelectric material properties**

The piezoelectric material properties are input in the material coordinate system. In this analysis, the material coordinate system coincides with the global system. Thus we have

Elastic constants: $E_1 = 61\times10^9$ Pa , $E_2 = 61\times10^9$ Pa , $E_3 = 53.2\times10^9$ Pa ,

$\nu_{12} = 0.35$ , $\nu_{13} = 0.38$ , $\nu_{23} = 0.38$ ,

$G_{12} = 22.593\times10^9$ Pa , $G_{13} = 21.1\times10^9$ Pa , $G_{23} = 21.1\times10^9$ Pa

(where 1=$x$, 2=$y$, 3=$z$, 12=$xy$, 13=$xz$, 23=$yz$)

Coupling constants (N/Vm): $e_{13} = -7.209$ , $e_{23} = -7.209$ , $e_{33} = 15.118$ ,
$e_{51} = 12.332$ , $e_{62} = 12.332$
(where 1=*x*, 2=*y*, 3=*z*, 4=*xy*, 5=*xz*, 6=*yz*)

Dielectric constants (C/Vm): $\varepsilon_{11} = 1.53 \times 10^{-8}$ , $\varepsilon_{22} = 1.53 \times 10^{-8}$ , $\varepsilon_{33} = 1.5 \times 10^{-8}$ .

Click the Manage Materials icon and click the Piezoelectric button to open the Define Piezoelectric Material dialog box, and add Material Number 2. Set "Input Elastic Constants As" to Modulus.

Fill in the Elastic Modulus Constants table as follows:

| | | |
|---|---|---|
| E1=61E9 | E2=61E9 | E3=53.2E9 |
| NU12=0.35 | NU13=0.38 | NU23=0.38 |
| G12=22.593E9 | G13=21.1E9 | G23=21.1E9 |

Fill in the Piezoelectric Coupling Constants table as follows:

| | k=1 | k=2 | k=3 |
|---|---|---|---|
| j=1 | | | -7.209 |
| j=2 | | | -7.209 |
| j=3 | | | 15.118 |
| j=4 | | | |
| j=5 | 12.332 | | |
| j=6 | | 12.332 | |

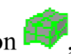
and fill in the Dielectric Constants table as follows:

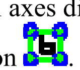
| | k=1 | k=2 | k=3 |
|---|---|---|---|
| j=1 | 1.53E-8 | | |
| j=2 | | 1.53E-8 | |
| j=3 | | | 1.5E-8 |

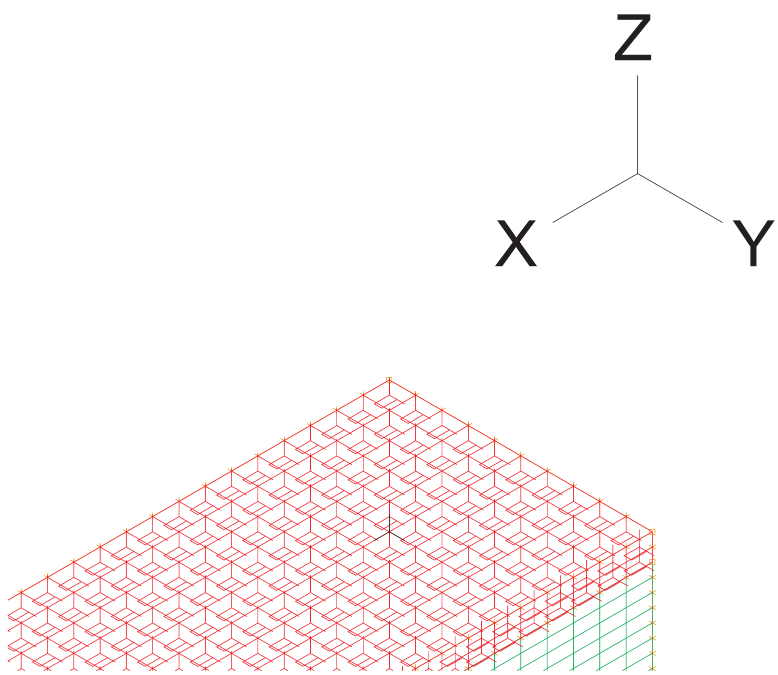
Click OK to close the dialog box and click Close to close the Manage Material Definitions dialog box.

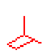
**Piezoelectric element group**

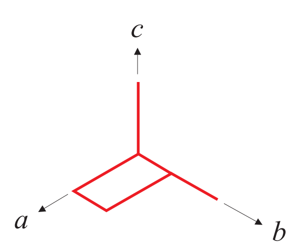
*Element group definition*: Click the Element Groups icon , add element group 2, set the Type to 3-D Solid, set the Element Option to Piezoelectric, set the Default Material to 2 and click OK.

*Mesh generation*: Click the Mesh Volumes icon , set the Element Group to 2, set the Nodes per Element to 8, enter 2 in the first row of the table and click OK.

*Material axis directions:* We would like to make sure that the material axes directions coincide with the global axes directions. Click the Color Element Groups icon  and the Show Material Axes icon . When you zoom to the upper-right region of the graphics window using the Zoom icon , the graphics window should look something like this:




The triad  drawn in each element shows the material axes of the element. The following diagram shows the correspondence of axes on this triad and the material axes *a*, *b* and *c*.



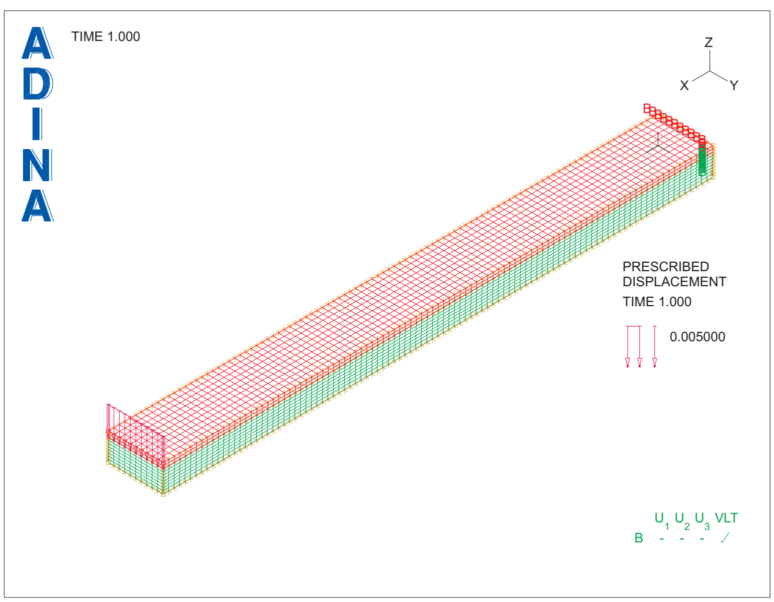
It is apparent that the material axes directions coincide with the global axes directions. Click the Unzoom All icon to see the entire model and click the Show Material Axes icon to hide the material axes triads.

**Boundary conditions and loading**

*Structural boundary conditions and loading:* We have prepared a batch file (`prob59_2.in`) that defines the following items:

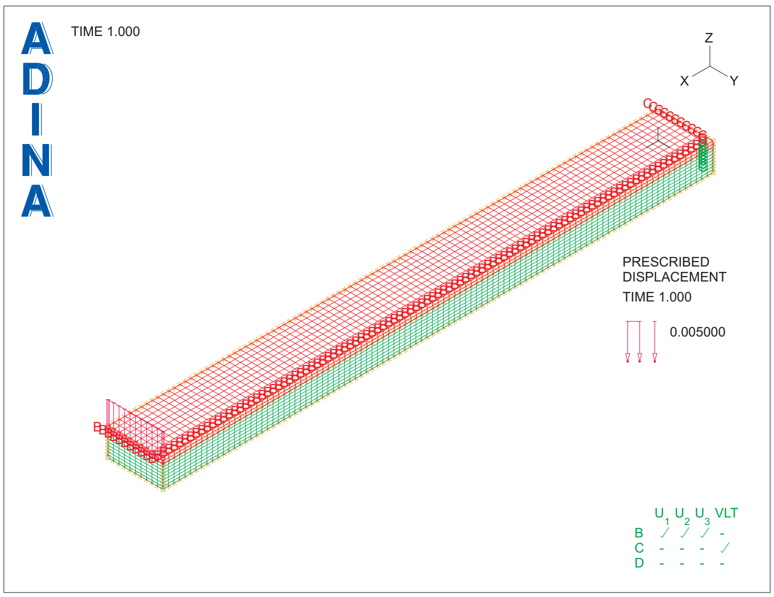
- ► Fixity of x, y and z-translations.
- ► Application of the above fixity to the model
- ► Displacement load.
- ► Application of the displacement load to the model.

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob59_2.in` and click Open. The graphics window should look something like this:




*Piezoelectric boundary condition:* Click the Apply Fixity icon, click the Define... button, add fixity VOLTAGE, check the Voltage field and click OK. In the Apply Fixity dialog box, set the "Fixity" to VOLTAGE, set the "Apply to" field to Face/Surface, enter 1 in the first row and column of the table and click OK.

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




Notice that the nodes on the boundary between the two layers are marked with a B, and that the B boundary condition has displacements free and voltage fixed.

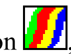
**Output of electromagnetic results to porthole file**

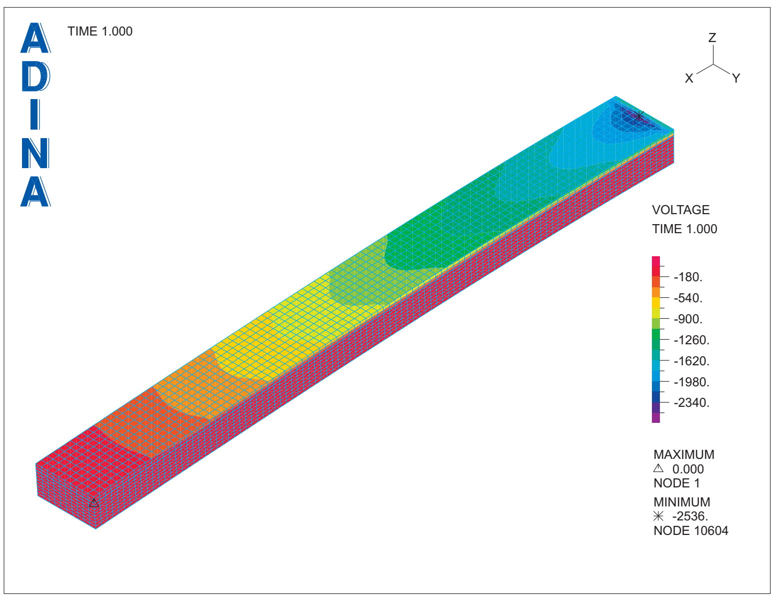
Choose Control→Porthole (.por)→Select Element Results, add Result Selection 1, set the Electromagnetic field to All and click OK.

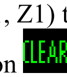
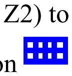
**Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file**

Click the Save icon and save the database to file `prob59`. Click the Data File/Solution icon, set the file name to `prob59a`, make sure that the Run Solution button is checked and click Save. 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 `prob59a`.

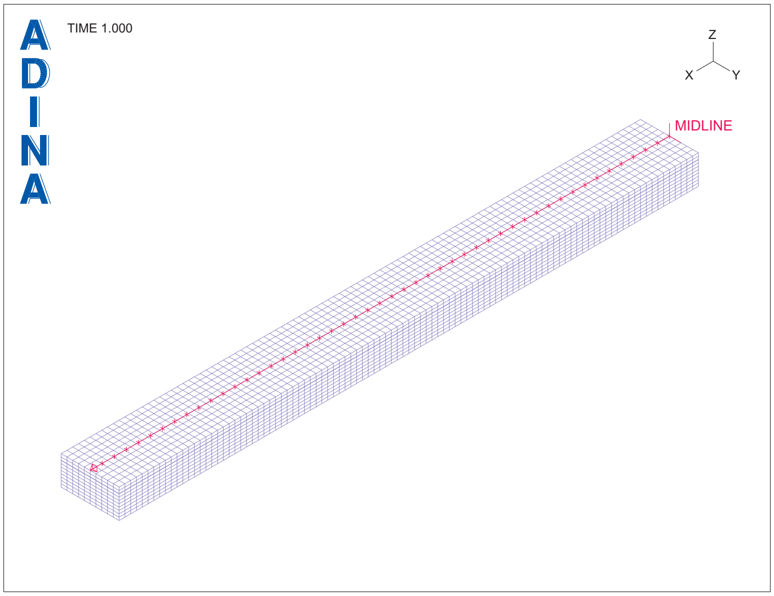
**Post-processing**

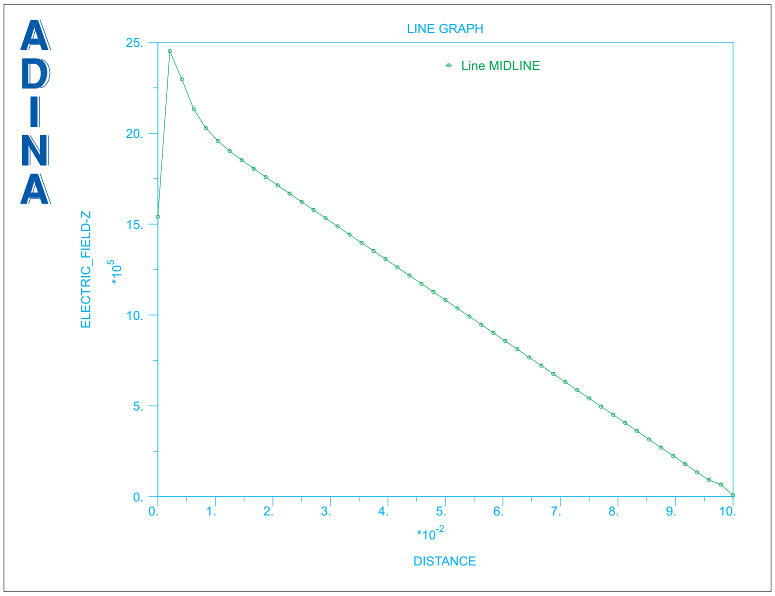
*Voltage band plot*: Click the Create Band Plot icon , set the Variable to (Electromagnetic: Voltage) and click OK. Move the band plot legends until the graphics window looks something like this:




*Electric field graph*: We would like to graph the electric field along a line within the model. Choose Definitions→Model Line→Stress Classification Line, add line MIDLINE, set (X1, Y1, Z1) to (0, 0.005, 0.005), (X2, Y2, Z2) to (0.1, 0.005,0.005) and click OK. Click the Clear icon , the Show Original Mesh icon  and the Show Deformed Mesh icon  (to hide the deformed mesh). Choose Display→Result Line Plot→Create and click OK. The graphics window should look something like the top figure on the next page:

Click the Clear icon , choose Graph→Response Curve (Model Line), make sure that the Model Line Name is MIDLINE, set the Y Coordinate Variable to (Electromagnetic: ELECTRIC_FIELD-Z) and click OK. The graphics window should look something like the bottom figure on the next page.

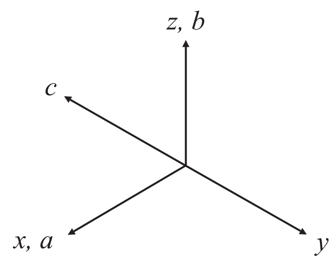



Choose Graph→List and confirm that, at distance 2.08333E-03, the maximum value of ELECTRIC_FIELD-Z is 2.45236E+06.

## Sensing analysis 2

Now we will repeat the analysis, changing the material axes so that the material axes directions do not coincide with the global axes directions. In this analysis, the material axes are oriented as follows:




The piezoelectric polarization is along the material b-direction.

Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file `prob59.idb` from the recent file list near the bottom of the File menu.

*Material constants:* The material constants need to be transformed into the material system. In order to keep track of the constants expressed in the different coordinate systems, we put an overbar over the constants expressed in the global system. The calculations and results are

Elastic constants: $E_1 = \overline{E}_x = 61\times10^9$ Pa , $E_2 = \overline{E}_z = 53.2\times10^9$ Pa , $E_3 = \overline{E}_y = 61\times10^9$ Pa ,

$\nu_{12} = \overline{\nu}_{xz} = 0.38$ , $\nu_{13} = \overline{\nu}_{xy} = 0.35$ , $\nu_{23} = \overline{\nu}_{zy} = \overline{\nu}_{yz}\dfrac{\overline{E}_y}{\overline{E}_z} = 0.43571$ ,

$G_{12} = \overline{G}_{xz} = 21.1\times10^9$ Pa , $G_{13} = \overline{G}_{xy} = 22.593\times10^9$ Pa ,

$G_{23} = \overline{G}_{zy} = \overline{G}_{yz} = 21.1\times10^9$ Pa

(where 1=*a*, 2=*b*, 3=*c*, 12=*ab*, 13=*ac*, 23=*bc*)

Coupling constants (N/Vm): $e_{12} = \overline{e}_{13} = -7.209$ , $e_{22} = \overline{e}_{33} = 15.118$ , $e_{32} = \overline{e}_{23} = -7.209$ ,

$e_{41} = \overline{e}_{51} = 12.332$ , $e_{63} = \overline{e}_{62} = 12.332$

(where 1=*a*, 2=*b*, 3=*c*, 4=*ab*, 5=*ac*, 6=*bc* in the unbarred quantities; 1=*x*, 2=*y*, 3=*z*, 4=*xy*, 5=*xz*, 6=*yz* in the overbarred quantities)

Dielectric constants (C/Vm): $\varepsilon_{11} = \overline{\varepsilon}_{11} = 1.53\times10^{-8}$ , $\varepsilon_{22} = \overline{\varepsilon}_{33} = 1.5\times10^{-8}$ ,

$\varepsilon_{33} = \overline{\varepsilon}_{22} = 1.53\times10^{-8}$ .

We have prepared a batch file (`prob59_3.in`) to automatically redefine the piezoelectric material. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob59_3.in` and click Open. Click the Manage Materials icon and click the Piezoelectric button. The Define Piezoelectric Material dialog box should show the following information:

Elastic Modulus Constants:

| | | |
|---|---|---|
| E1=61E9 | E2=53.2E9 | E3=61E9 |
| NU12=0.38 | NU13=0.35 | NU23=0.43571 |
| G12=21.1E9 | G13=22.593E9 | G23=21.1E9 |

Piezoelectric Coupling Constants:

| | k=1 | k=2 | k=3 |
|---|---|---|---|
| j=1 | | -7.209 | |
| j=2 | | 15.118 | |
| j=3 | | -7.209 | |
| j=4 | 12.332 | | |
| j=5 | | | |
| j=6 | | | 12.332 |

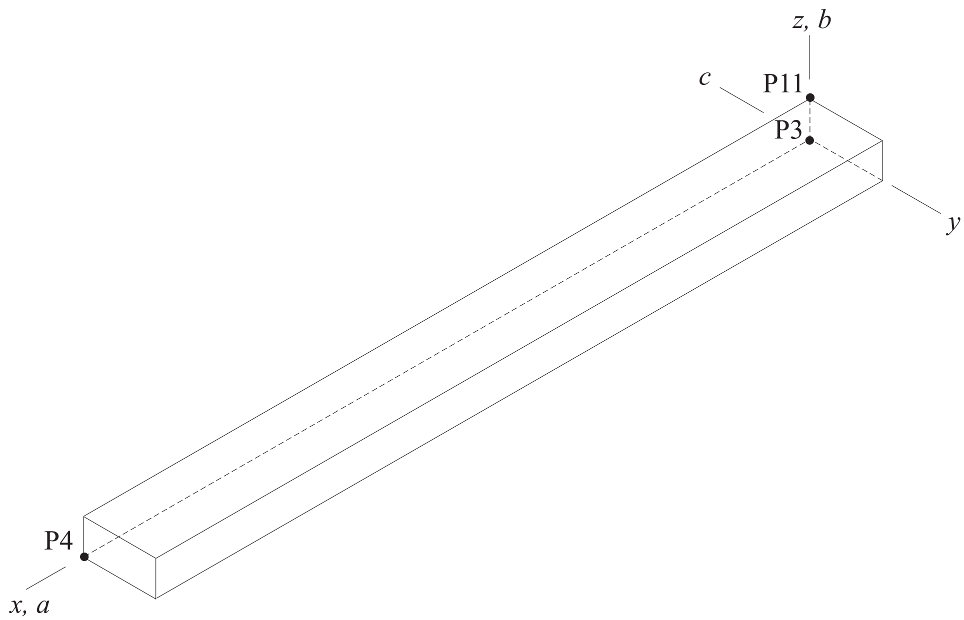
Dielectric Constants:

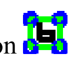
| | k=1 | k=2 | k=3 |
|---|---|---|---|
| j=1 | 1.53E-8 | | |
| j=2 | | 1.5E-8 | |
| j=3 | | | 1.53E-8 |

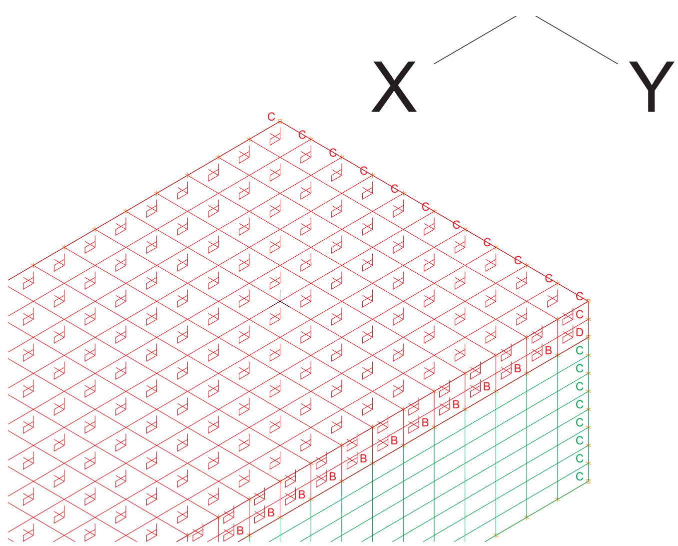
Click Cancel, then click Close, to close both dialog boxes.

*Material axes:* We now need to define the material axes. We will define the material axes using three geometry points, as shown in the figure on the next page.

Choose Model→Orthotropic Axes Systems→Define, add System Number 1, set the "Defined by" field to 3 Points, set Point 1 to 3, Point 2 to 4, Point 3 to 11 and click OK. Choose Model→Orthotropic Axes Systems→Assign (Material), and, in the Assign Material Axes System dialog box, make sure that the Axes System is set to 1. Then set the "Apply to" field to Volume and, in the first row of the table, set the Volume to 2, then click OK.

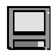



Click the Show Material Axes icon  to show the material axes. When you zoom to the upper-right region of the graphics window using the Zoom icon , the graphics window should look something like this:



We can see that the local *a*-direction is parallel to the global *x*-direction and that the local *b*-direction is parallel to the global *z*-direction.

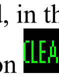
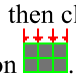
**Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file**

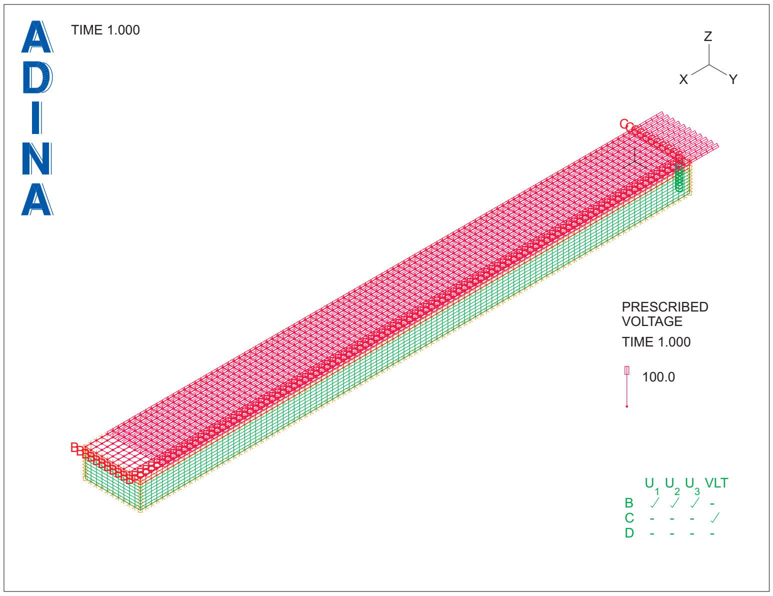
Click the Save icon  to save the database to file `prob59`. Click the Data File/Solution icon , set the file name to `prob59b`, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon  and open porthole file `prob59b`.

**Post-processing**

Repeat the post-processing steps given above. The results should be exactly the same as in the first analysis.

## Actuating simulation

Now we will change the loading from prescribed displacement to prescribed voltage and perform the actuating analysis. Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file `prob59.idb` from the recent file list near the bottom of the File menu. Click the Apply Load icon , set the Load Type to Displacement, set the "Apply to" field to Line, click Clear and then Apply. Set the Load Type to Voltage and click the Define... button, add Voltage Number 1, set the Magnitude to 100 and click OK. In the Apply Load dialog box, set the "Apply to" field to Surface, and, in the first row of the table, set the Surface number to 4, then click OK. Click the Clear icon , the Boundary Plot icon  and the Load Plot icon . The graphics window should look something like the figure on the next page.

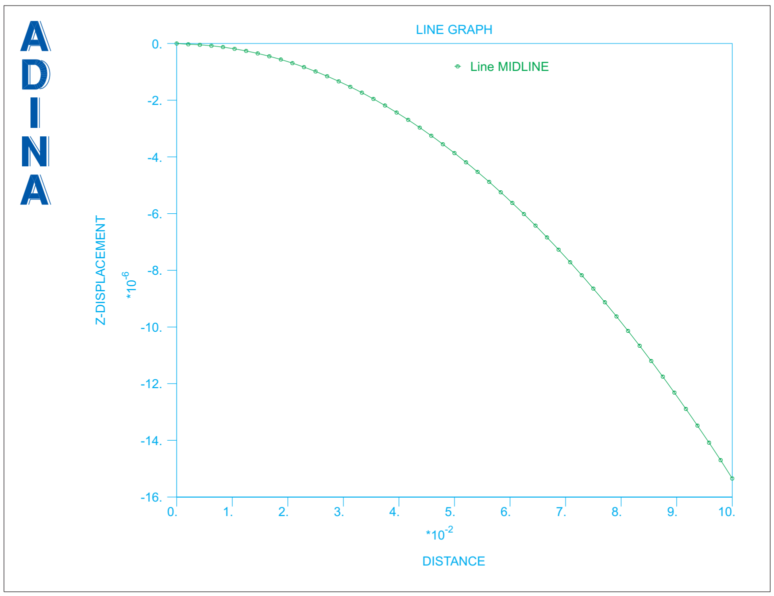



**Generating the ADINA Structures data file, running ADINA Structures, loading the porthole file**

Click the Save icon to save the database to file `prob59`. Click the Data File/Solution icon, set the file name to `prob59c`, make sure that the Run Solution button is checked and click Save. 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 `prob59c`.

**Post-processing**

Choose Definitions→Model Line→Stress Classification Line, add line MIDLINE, set (X1, Y1, Z1) to (0, 0.005, 0.005), (X2, Y2, Z2) to (0.1, 0.005,0.005) and click OK. Click the Clear icon, choose Graph→Response Curve (Model Line), make sure that the Model Line is set to MIDLINE, set the Y Coordinate Variable to (Displacement: Z-DISPLACEMENT) and click OK. The graphics window should look something like the figure on the next page.




Choose Graph→List and confirm that at distance 1.00000E-01, the z-displacement has the value -1.53446E-05.

**Exiting the AUI**

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

This page intentionally left blank.