

Task 5

Static deformation of a piezoceramic transducer with multi-electrode coating

Objectives of the assignment:

1. Study the main features of solving the problems of electroelasticity in ANSYS APDL:

- setting the parameters of piezoelectric material
- defining the element coordinate systems and using them to set anisotropic and piezoelectric properties of the material
- using mapped finite element mesh
- defining the electrode surfaces and setting the boundary conditions on them

The example problem is a two-dimensional static problem for a piezoelectric transducer with inhomogeneous polarization and multielectrode covering.

2. Write a program in ANSYS APDL for an individual assignment problem (static problem for a piezoelectric transducer with inhomogeneous polarization and multielectrode covering)

3. Perform computations, analyze the results and prepare a report.

Example problem

The piezoelectric disk of the radius R and the thickness H is assumed to be in a state of axisymmetric deformation in a cylindrical coordinate system Orz ($0 \leq r \leq R, -H/2 \leq z \leq H/2$). By virtue of axial symmetry, it is enough to consider only a meridional section of the disk. In accordance with ANSYS methodology, r is the X -axis, and z is the Y -axis.

The disk has four electrode surfaces (Fig. 1): electrode 1 is $0 \leq X \leq R_1, Y = -H/2$; electrode 2 is $0 \leq X \leq R_1, Y = H/2$; electrode 3 is $R_2 \leq X \leq R, Y = -H/2$; electrode 4 is $R_2 \leq X \leq R, Y = H/2$. The disk is made of piezoceramics PZT-4. Two of its zones $\{0 \leq X \leq R_1, -H/2 \leq Y \leq H/2\}$ and $\{R_1 \leq X \leq R_2, -H/2 \leq Y \leq H/2\}$ are polarized along the axis Y , and the zone $\{R_2 \leq X \leq R, -H/2 \leq Y \leq H/2\}$ is polarized opposite to the direction of the axis Y (Fig. 2).

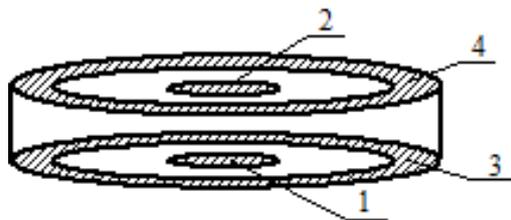


Fig. 1 Sketch of piezoelectric disk with four electrodes

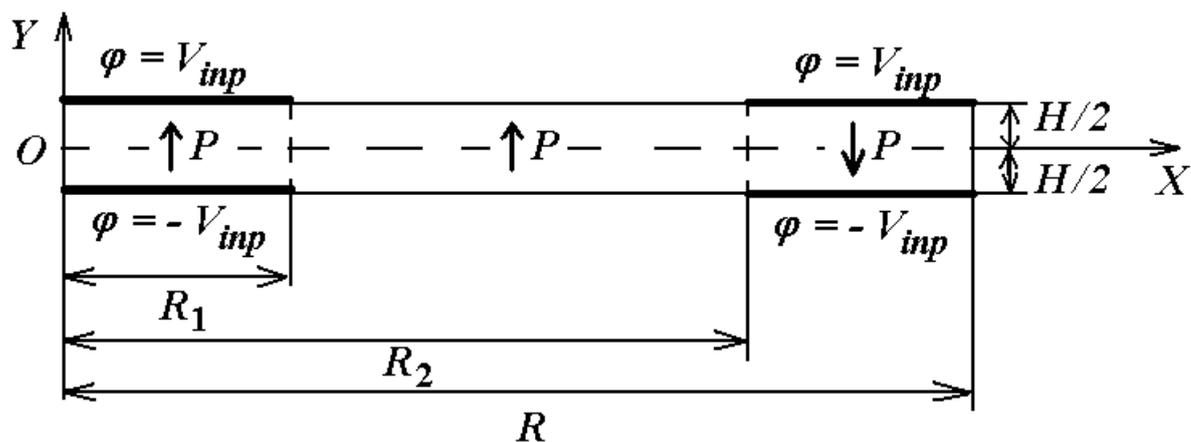


Fig. 2 Meridional section of a piezoelectric disk

The disk is fixed in Y direction in the middle points of its outer surface, i.e.. $U_y = 0$ при $X = R, Y = 0$, the symmetry conditions hold at $X = 0$, and the rest of the boundaries are free from mechanical stresses.

The disk is deformed by the applied electric voltage to the electrodes. The boundary conditions are: $\varphi = -V_{inp}$ on the first electrode, $\varphi = V_{inp}$ on the second electrode, $\varphi = -V_{inp}$ on the third electrode, and $\varphi = V_{inp}$ on the fourth electrode.

The objective of the problem is to determine the displacements of the disk caused by its deformation.

SOLVING THE PROBLEM USING ANSYS

The example of problem solving using ANSYS is provided in the file **Piezodisk_St.inp**. The example of problem solving using ANSYS is provided in the file: Coupled-Field Guide, 2.14. Sample Piezoelectric Analysis (Batch or Command Method), 2.16. Sample Electroelastic Analysis of a Dielectric Elastomer (Batch or Command Method); Verification Manual, файлы Vm175.dat, Vm176.dat, Vm231.dat, Vm237.dat (ссылки приведены для ANSYS 11.0).

ANSYS Product Launcher is the best option to start working with ANSYS. In ANSYS Product Launcher, select *Simulation Environment*→ANSYS, choose *Working Directory* for storing the working files and define *Job Name* for the project.

If there is an existing ANSYS database, then after the launch of ANSYS it can be resumed from File→Resume.

An ANSYS command file (text file with extension .inp, .dat or.txt), written in APDL ANSYS, can be executed from File → Read Input from... When creating a command file, it is useful to copy the commands from the input file into the command line, execute them step by step and look at the results in interactive mode.

It is recommended to save results from time to time! The current database should be cleared before performing new analysis File→Clear and Start New.

Definition of material constants for piezoelectric materials

As it is known, the piezoelectric effect can be observed only in crystals that do not have the central symmetry. Therefore, the piezoelectric bodies must have

anisotropic properties. In a general case, in order to set the material constants of piezoelectric bodies in ANSYS, the following values should be defined: the density ρ ; the symmetric matrix of elastic moduli $c_{\alpha\beta}^E$; $\alpha, \beta = 1, \dots, 6$; $c_{\alpha\beta}^E = c_{\beta\alpha}^E$; the matrix of piezoelectric modules $e_{i\alpha}$; $i = 1, 2, 3$; $\alpha = 1, \dots, 6$; and the diagonal matrix of dielectric permittivities ϵ_{ii}^S ; $i = 1, 2, 3$. (It is possible to set alternative sets of material moduli such as elastic compliances $s_{\alpha\beta}^E$ and so on)

Let us describe the method of defining the modules $c_{\alpha\beta}^E$ and $e_{i\alpha}$ in ANSYS in more detail. The coefficients $c_{\alpha\beta}^E$ are set in a form of 6x6 matrix (4x4 for 2D problems). By virtue of symmetry, only the upper triangular part of the matrix of elastic moduli is used, and the coefficients of the matrix are arranged in a specific way:

$$\mathbf{c}_{\text{ANSYS,3D}} = \begin{array}{c|cccccc} & x & y & z & xy & yz & xz \\ \hline x & c_{11}^E & c_{12}^E & c_{13}^E & c_{16}^E & c_{14}^E & c_{15}^E \\ y & & c_{22}^E & c_{23}^E & c_{26}^E & c_{24}^E & c_{25}^E \\ z & & & c_{33}^E & c_{36}^E & c_{34}^E & c_{35}^E \\ xy & & & & c_{66}^E & c_{46}^E & c_{56}^E \\ yz & & \text{sym} & & & c_{44}^E & c_{45}^E \\ xz & & & & & & c_{55}^E \end{array} \quad (1.1)$$

$$\mathbf{c}_{\text{ANSYS,2D}} = \begin{array}{c|cccc} & x & y & z & xy \\ \hline x & c_{11}^E & c_{12}^E & c_{13}^E & c_{16}^E \\ y & & c_{22}^E & c_{23}^E & c_{26}^E \\ z & & & c_{33}^E & c_{36}^E \\ xy & & \text{sym} & & c_{66}^E \end{array} \quad (1.2)$$

where $c_{\alpha\beta}^E$ define elastic moduli in the theory of piezoelectricity.

The components of the matrix $\mathbf{c}_{\text{ANSYS,3(2)D}}$ in ANSYS are defined by the commands: `TB,ANISO,MAT` and `TBDATA,STLOC,C1,C2,...,C6`; where `MAT` is the number of the material properties set, `STLOC` is the starting location in the data table for entering data values `C1,C2,...,C6` in 1D array. The data table for the coefficients that enter in (1.1), (1.2), is filled by rows (1.1) in a form of one-dimensional array with 21 components:

$$\begin{bmatrix} 1 & 2 & 3 & 4 & 5 & 6 \\ & 7 & 8 & 9 & 10 & 11 \\ & & 12 & 13 & 14 & 15 \\ & & & 16 & 17 & 18 \\ & & & & 19 & 20 \\ & & & & & 21 \end{bmatrix} \quad (1.3)$$

The resulting correspondence between the data from the data table defined by TBDATA, command and the elasticity moduli $c_{\alpha\beta}^E$:

N $\underline{0}$	1	2	3	4	5	6	7	8	9	10	
$c_{\alpha\beta}^E$	c_{11}^E	c_{12}^E	c_{13}^E	c_{16}^E	c_{14}^E	c_{15}^E	c_{22}^E	c_{23}^E	c_{26}^E	c_{24}^E	
N $\underline{0}$	11	12	13	14	15	16	17	18	19	20	21
	c_{25}^E	c_{33}^E	c_{36}^E	c_{34}^E	c_{35}^E	c_{66}^E	c_{46}^E	c_{56}^E	c_{44}^E	c_{45}^E	c_{55}^E

The most common type of piezoelectric materials is the piezoceramics polarized along the axis Oz , whose nonzero elastic moduli in TBDATA data table will have the following numbers: (for piezoceramic $c_{22}^E = c_{11}^E$; $c_{23}^E = c_{13}^E$; $c_{55}^E = c_{44}^E$; $c_{66}^E = (c_{11}^E - c_{12}^E)/2$):

N $\underline{0}$	1	2	3	7	8	12	16	19	21
$c_{\alpha\beta}^E$, piezoceramic	c_{11}^E	c_{12}^E	c_{13}^E	c_{11}^E	c_{13}^E	c_{33}^E	c_{66}^E	c_{44}^E	c_{44}^E

As it can be seen from comparison of (1.1) – (1.3), for 2D problems it is enough to specify the first 16 positions in a one-dimensional array of the moduli $c_{\alpha\beta}^E$.

We note that for plane and axisymmetric 2D problems the plane Oxy is the working plane, and the axis Oy is usually considered to be the axis of polarization (axis $z=(3)$ for the moduli $c_{\alpha\beta}^E$). In this case for plane and axisymmetric problems different nonzero elastic moduli for piezoceramics in the data table TBDATA will have the following numbers:

N $\underline{0}$	1	2	3	7	8	12	16
$c_{\alpha\beta}^E$, piezoceramic	c_{11}^E	c_{13}^E	c_{12}^E	c_{33}^E	c_{13}^E	c_{11}^E	c_{44}^E

In a similar unusual way the piezomoduli $e_{i\alpha}$ are set in ANSYS. The piezomoduli are arranged in a 6x3 (4x2 for 2D problems):

$$\mathbf{e}_{\text{ANSYS,3D}} = \begin{array}{c|ccc} & x & y & z \\ \hline x & e_{11} & e_{21} & e_{31} \\ y & e_{12} & e_{22} & e_{32} \\ z & e_{13} & e_{23} & e_{33} \\ xy & e_{16} & e_{26} & e_{36} \\ yz & e_{14} & e_{24} & e_{34} \\ xz & e_{15} & e_{25} & e_{35} \end{array}; \quad \mathbf{e}_{\text{ANSYS,2D}} = \begin{array}{c|cc} & x & y \\ \hline x & e_{11} & e_{21} \\ y & e_{12} & e_{22} \\ z & e_{13} & e_{23} \\ xy & e_{16} & e_{26} \end{array}$$

After the execution of the command **TB,PIEZ,MAT** the data table TBDATA for the piezomoduli will be filled by rows with the values from the matrix $\mathbf{e}_{\text{ANSYS,3D}}$ as a one-dimensional array of the size 18. As a result, we obtain the correspondence between the data from the data table TBDATA and the piezomoduli $e_{i\alpha}$:

N _o	1	2	3	4	5	6	7	8	9
$e_{i\alpha}$	e_{11}	e_{21}	e_{31}	e_{12}	e_{22}	e_{32}	e_{13}	e_{23}	e_{33}
N _o	10	11	12	13	14	15	16	17	18
$e_{i\alpha}$	e_{16}	e_{26}	e_{36}	e_{14}	e_{24}	e_{34}	e_{15}	e_{25}	e_{35}

where for 2D problems it is enough to define the elements with numbers 1, 2, 4, 5, 7, 8, 10 and 11.

For the piezoceramics polarized along the direction of the axis Oz , the nonzero piezomoduli of piezoceramic material for 3D problems in the data table TBDATA will have the following numbers (for piezoceramic $e_{32} = e_{31}$; $e_{24} = e_{15}$):

N _o	3	6	9	14	16
$e_{i\alpha}$, piezoceramic	e_{31}	e_{31}	e_{33}	e_{15}	e_{15}

Finally, for plane and axisymmetric problems in the cases, when in the plane Oxy the axis Oy is the axis of preliminary polarization of piezoceramics, and different nonzero piezomoduli in the data table TBDATA will have the numbers that are indicated in the table:

N _o	2	5	8	10
$e_{i\alpha}$, piezoceramic	e_{31}	e_{33}	e_{31}	e_{15}

Summarizing the above, for the piezoceramic material we can write the following fragments of program code in ANSYS APDL, that define the sets of material constants *MAT* with number 1 for 3D and 2D problems, where for the latter

case the piezoceramics is considered to be polarized along the Oy -axis of the working plane Oxy :

Fragment 1 (3D).

```
MP, DENS, 1, RHO
TB, ANEL, 1
TBDATA, 1, C11E, C12E, C13E
TBDATA, 7, C11E, C13E
TBDATA, 12, C33E
TBDATA, 16, C66E
TBDATA, 19, C44E
TBDATA, 21, C44E
TB, PIEZ, 1
TBDATA, 3, E31
TBDATA, 6, E31
TBDATA, 9, E33
TBDATA, 14, E15
TBDATA, 16, E15
MP, PERX, 1, EPS11
MP, PERZ, 1, EPS33
```

Fragment 2 (2D).

```
MP, DENS, 1, RHO
TB, ANEL, 1
TBDATA, 1, C11E, C13E, C12E
TBDATA, 7, C33E, C13E
TBDATA, 12, C11E
TBDATA, 16, C44E
TB, PIEZ, 1
TBDATA, 2, E31
TBDATA, 5, E33
TBDATA, 8, E31
TBDATA, 10, E15
MP, PERX, 1, EPS11
MP, PERY, 1, EPS33
```

Here $RHO=\rho$; $C11E=c_{11}^E$ and so on. These values are scalar parameters in ANSYS APDL and their values should be defined beforehand. Besides, in the presented fragment of the program code we have added **MP** command, that defines the density and the dielectric permittivities of piezoceramics.

As it can be seen, the definition of piezoelectric materials requires some efforts. It should be emphasized that in the presented fragments the axes of $Oxy(z)$ -plane are the axes of the element coordinate systems for the defined constant sets.

Let us now return to the listing of the input file for solving the test 2D static problem for piezoelectric transducer with inhomogeneous polarization and multi-electrode coating.

The command **ET,1,PLANE223,1001,,1** defines the quadrilateral 8-node finite element PLANE223 with the options of piezoelectric analysis and axisymmetry. This element will be used to solve the problem. For ANSYS user all the distinctions between axisymmetric problem and 2D plane stress problem consist in the third option of this finite element (the command **ET,1,PLANE223,1001,,0** will define the element for 2D plane stress problem). However, with this option ANSYS will use the equations of axisymmetric theory of piezoelectricity that are much more complex than the equations of plane stress or plane strain.

Solid model of the meridional section of piezoelectric disk

In ANSYS APDL the solid model of the original domain with complex geometry is usually constructed from bottom to top', starting with the most simple

objects (Entities), which are the Keypoints, and finishing with Areas for 2D problems or Volumes for 3D problems.

K command is used to build the keypoints. Its first argument is the reference number of the keypoint, and the remaining arguments are the coordinates of the keypoint. For example, **K,2,R1,-H/2** command defines a keypoint with the reference number 2 and coordinates $x=R1$, $y=-H/2$, $z=0$ (the last coordinate takes the default value). The Line between two keypoints can be defined by **L** command. For example, **L,2,3** command defines a line in the active coordinate system between the keypoints 2 and 3. The lines are numbered automatically, starting with the lowest available number. In order to construct the areas, we can use the commands **A** and **AL**, where the resulting areas are also numbered automatically. **AL** command creates an area bounded by the previously defined lines. The lines (10 lines maximum) must be input in clockwise or counterclockwise order and must form a simply connected closed curve. **A** command defines an area by connecting the keypoints. The keypoints defining the area must be input in clockwise or counterclockwise order around the area (maximum 18 keypoints in the list). The existing lines between adjacent keypoints will be used; and the missing lines will be generated as straight lines in the active coordinate system and will be assigned the lowest available numbers.

According to Fig. 2, in the meridional section of the disk we can distinguish two zones with two different polarization directions. Therefore the axisymmetric model of the disk can be composed by two areas. Fig. 3 shows the areas A1 and A2 with indicated area numbers and keypoint numbers (Menu path Plot->Areas, to show the numbers of areas and keypoints select PlotCtrls->Numbering-> tick Area numbers, Keypoint numbers)

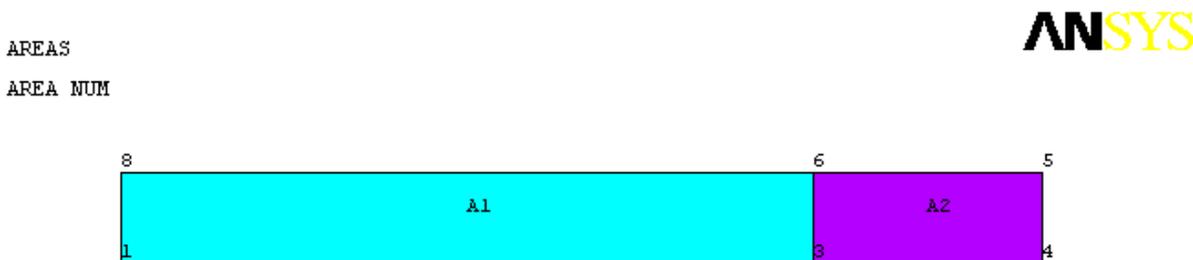


Fig. 3 Area numbering in the model of a meridional section of piezoelectric disk

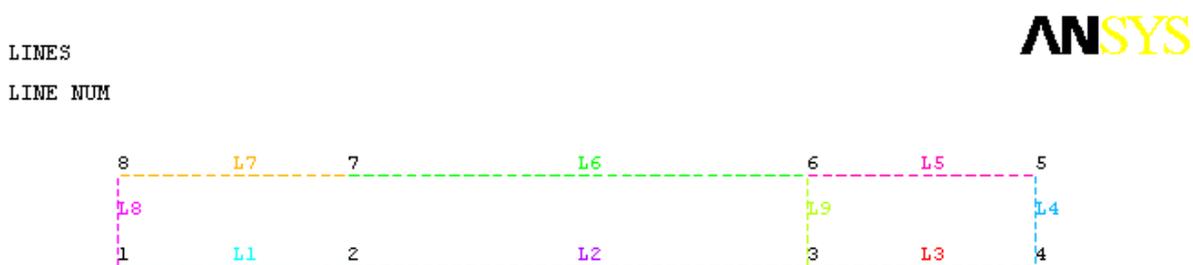


Fig. 4 Keypoints and meshed lines of the disk section model

The lines and the keypoints that constitute the areas A1 and A2 can be seen in Fig. 4 (Menu path: Plot->Lines (preliminary go to Select->Entities and select the lines L1-L9), show the numbers of lines and keypoints by selecting: PlotCtrls->Numbering->tick Line numbers, Keypoint numbers). Here a rectangular area A2 is constructed in a standard way of four lines L3, L4, L5, L9 and four keypoints 3, 4, 5, 6. The upper and the lower boundaries of this area coincide with the lines of the electrodes location. The area A1 consists of six keypoints 1, 2, 3, 6, 7, 8 and six lines L1, L2, L9, L6, L7, L8. For this area both upper and lower boundaries must be constructed of two lines, and the keypoints of the lines L1 and L7 that simulate the electrodes must coincide with the ends of the electrodes. This requirement ensures that the ends of the electrodes will coincide with the nodes of finite elements. It is also worth noting, that the line L9 is common for the adjacent areas A1 and A2.

Direction of the polarization vector for the areas of piezoelectric disk

For plane and axisymmetric 2D problems it is convenient to consider the axis Oy as the axis of preliminary polarization in the working plane Oxy , and therefore in ANSYS for the piezomaterials with different polarization vectors it is enough to define for the corresponding areas the element coordinate systems that are rotated in the proper way with respect to the global coordinate system.

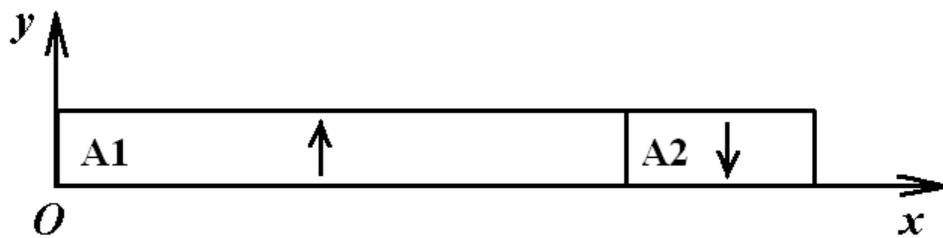


Fig. 5 The directions of the polarization vectors P for the areas of the disk section

For our example problem the direction of the polarization vector P in the area A1 coincides with the direction of the Oy -axis of the global Cartesian coordinate system, therefore for the finite elements of this area it is not necessary to introduce special coordinate system. For the area A2 the direction of the polarization vector P is opposite to the direction of the Oy -axis of the global Cartesian coordinate system, therefore for this area it is necessary to define the element coordinate system with the axis Oy' , directed along the polarization vector, i. e. the axis Oy' must be directed in the way opposite to the axis Oy .

To achieve this, we need to rotate the original element coordinate system to 180 degrees with respect to the global Cartesian coordinate system. The next block of commands creates the local coordinate system and defines it as an element coordinate system for finite elements that will be created in the area A2.

```

LOCAL,11,0,,,180
ASEL,S,AREA,,2
AATT,1,1,11

```

Fig. 6 shows an example of finite element mesh with the element coordinate systems (see green lines). To show finite element mesh, select Menu path: Plot -> Elements. To show polarization, select Menu path: PlotCtrls -> Numbering->Elem/Attrib numbering-> select Element CS num, Numbering shown with Colors only, to show the direction of the element coordinate system for every element select PlotCtrls->Symbols, tick ESYS Element coordinate sys)

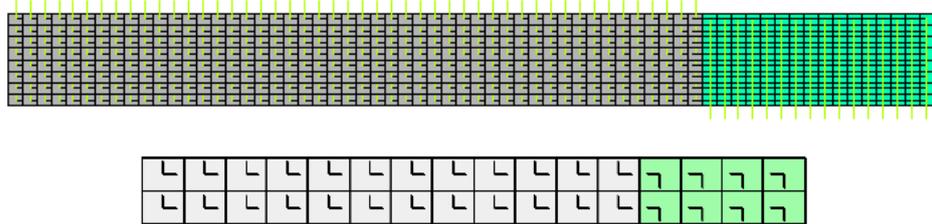


FIG. 6 Element coordinate systems in the areas of meridional section of the disk: general view and zoomed region

Finite element mapped mesh

As the element **PLANE223** is a quadrilateral finite element and the simulated area of the meridional section of the disk is a quadrangle, more precisely, rectangle, it is therefore logical to use **mapped** finite element meshing in quadrangles (or, even better, in rectangles).

There are two main meshing methods in ANSYS: Free Mesh and Mapped Mesh. A free mesh does not follow any pattern and has no element shape restrictions. Free mesh is suitable for areas and volumes of complex shapes. Free mesh is usually defined by an average finite element size. On the contrary, a mapped mesh restricts the element shape (for example, quadrangles for areas, or hexahedrons for volumes) and typically has a regular pattern with obvious rows of elements. Mapped mesh is beneficial for areas and volumes of simple shape, such as rectangles or bricks. The use of a mapped mesh facilitates to reduce the computation time. Usually a mapped mesh is built on the base of the divisions and spacing ratio, which are specified for its constituting lines. Generally, a mapped mesh is more precise than a free mesh with comparable number of elements and nodes.

In order to build a mapped quadrilateral finite element mesh for a quadrilateral area, this area must satisfy two conditions: 1) the area must consist of four lines, 2) the area must have equal numbers of element divisions specified on the opposite sides. If an area is bounded by more than four lines, then some of the lines can be combined (by **LCOB** command) or concatenated (by **LCCAT** command) in order to reduce the total number of lines to four.

In our example problem there are at least two ways to construct a solid model of the meridional section of the disk that would accept mapped finite element mesh. For example, we could build an additional line between keypoints 2 and 7 and then create three rectangular areas, each of them consisting of four lines.

Here we use another way that allows us to reduce the number of the required areas to two. We create two areas A1 and A2, where A2 is a regular area that accepts a mapped mesh, and A1 is irregular in the way that it does not accept a mapped mesh because it consists of six lines. In order to make area A1 regular, it is enough to

concatenate lines L1 and L2 which constitute the lower boundary and lines L6 and L7 which constitute the upper boundary. This can be done by **LCCAT**, command which concatenates multiple, adjacent lines into one line in preparation for mapped meshing. Note that **LCCAT** accepts only two arguments, so if it is necessary to concatenate more than two lines, the first argument should be set to **ALL**. In this case the second argument is ignored and all previously selected lines (by **LSEL** command) are concatenated. It is also worth noting that for a mapped mesh the total number of the line divisions must be the same for the opposite lines of the area. Fig.7 shows the result of concatenation for the lines of the upper and lower boundaries of the area A1. The output lines L10 and L11 keep the element divisions of the input lines.

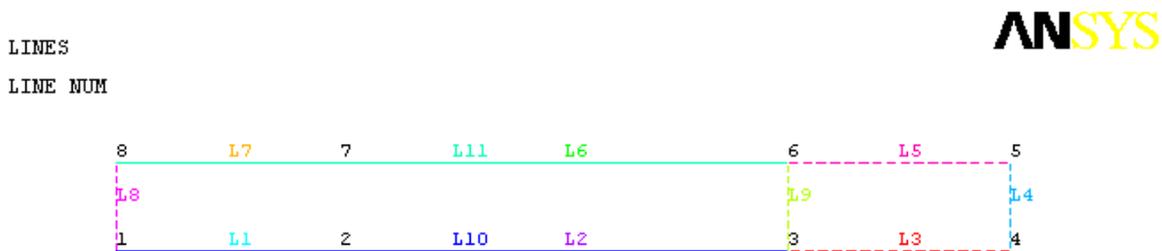


Fig. 7 Line concatenation for a solid model of the meridional section of the disk

Boundary conditions

1) Boundary conditions on the electrode surfaces.

In the theory of piezoelectricity the piezoelectric materials are considered to be active dielectrics. The parts of the boundary Γ_{V_m} ; $m=1,2,\dots,M$; can be represented by the metallized surfaces or electrodes. The electrodes Γ_{V_m} are assumed to be equipotential surfaces, i. e. the electric potential $\varphi(\mathbf{x},t)$ on these surfaces should not depend on the coordinate vector \mathbf{x} :

$$\varphi = V_m(t); \quad \mathbf{x} \in \Gamma_{V_m} \quad (2.1)$$

The boundary conditions (2.1) are called essential or kinematic electric boundary conditions for the boundary-value problems of piezoelectricity, if $V_m(t)$ are known functions. The electrodes with known $V_m(t)$ are called the electrodes powered by voltage generators, and when $V_m = 0$ they are called short-circuited grounded electrodes.

The other type of electrodes are the electrodes Γ_{Q_l} , powered by current generators I_l . For the electrodes Γ_{Q_l} ; $l=1,2,\dots,L$; the boundary conditions have the form:

$$\varphi = \Phi_l(t); \quad \mathbf{x} \in \Gamma_{Q_l} \quad (2.2)$$

$$\int_{\Gamma_{Q_l}} \mathbf{n} \cdot [\mathbf{D}] d\Gamma = Q_l; \quad \dot{Q}_l = \pm I_l$$

where \mathbf{n} is the normal to Γ_{Q_l} ; $[\mathbf{D}]$ is the jump of the electric induction vector \mathbf{D} ; Q_l is the total surface charge on the electrode Γ_{Q_l} ; where the values of the electric

potentials $\Phi_l(t)$ are initially unknown. When $Q_l = 0$, the electrodes (2.2) are called open or passive electrodes.

Boundary conditions (2.1) and (2.2) can be implemented in ANSYS in the following way. The nodes of the finite element mesh that belong to one electrode Γ_{V_m} or Γ_{Q_l} can be combined in one node (coupled DOF) by **CP** command that will be a reference node for this electrode. Then for this reference node N_VOLT we can either set the value of the electric potential $VM=V_m$ by the command **D,N_VOLT,VOLT,VM**, or define the total electric charge $QL=Q_l$ by the command **F,N_VOLT,AMPS,QJ** (for finite elements PLANE223, SOLID226, SOLID227) or by the command **F,N_VOLT,AMPS,QJ** (for finite elements K \in PLANE13, SOLID5, SOLID98). The block of commands, that define a group of nodes and set the value of the electric potential on the first electrode, is presented below:

```
NSEL,S,LOC,Y,-H/2
NSEL,R,LOC,X,0,R1
CP,1,VOLT,ALL
*GET,N_VOLT1,NODE,,NUM,MIN
D,N_VOLT1,VOLT,-VINP
```

In order to define a free electrode, it is enough to define a group of nodes by **CP** command with using **D** command.

2) Mechanical boundary conditions. In our example problem we need to set the symmetry boundary conditions with respect to the axis of rotation Oy and the constraint on the displacements along Oy for the middle point on the outer surface radius of the meridional section of the disk.

The degrees-of-freedom constraints (three DOFs UX, UY, VOLT for the element PLANE223 with the options of piezoelectric analysis KEYOPT(1) = 1001) can be set either by **D** command that defines DOF constraints at nodes or by **DL** command that defines DOF constraints on lines. It should be noted that all solid boundary conditions will be transformed into finite element boundary conditions at the stage of the solution. The solid boundary conditions set on lines have priority over the finite element boundary conditions for the nodes on the same line.

2.1) **The condition of symmetry** can be set either by **DL** command or by **D** command. **D** command accepts a symmetry option SYMM, as in the above example **DL,8,,SYMM**. Another way is to select all necessary nodes and use **D** command. We can select the nodes that lie on the line L8 by the following commands:

```
LSEL,S,LINE,,8
NSLL,S,1 ! Select nodes belonging to selected lines, including line ends
! second argument is the key to nodes selection: 0 – select only internal nodes
(default value), 1 – select all nodes interior to line and at keypoints.
```

D,ALL,UX,0 command assigns zero UX displacements for all selected nodes, and this corresponds to the symmetry with respect to the axis Oy . In a similar way the command **D,ALL,UY,0** will set zero displacements UY, which corresponds to the symmetry with respect to the axis Ox .

We note that in the considered axisymmetric problem, as it can be easily checked in ANSYS, the symmetry boundary conditions on the line $x = 0$ will be satisfied automatically, and the above command **DL,8,,SYMM** is provided for reliability and is therefore optional.

2.2) **The condition of a rigidly fixed boundary** assumes that for the given nodes the displacement vector is equal to zero, i. e. in a case of plane problem $UX=0$ and $UY=0$. The corresponding block of commands can be written as:

D,ALL,UX,0

D,ALL,UY,0

Note that in our example problem we use only a constraint on the displacements along Oy -axis, i. e. $UY=0$.

Finite element model of piezoelectric disk with boundary conditions is shown in Fig. 8 (Menu path Plot->Elements, to show boundary conditions select PltCtrls->Symbols->and tick “All applied BC”.)

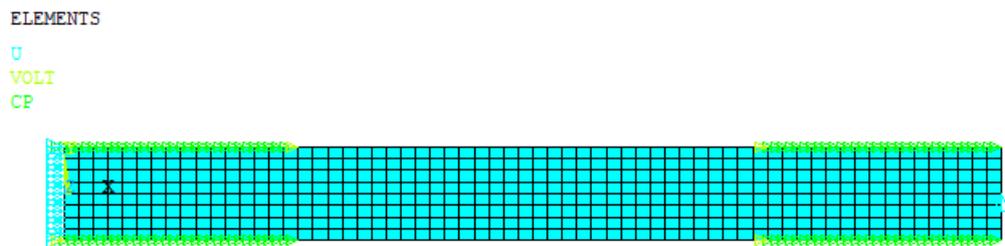


Fig. 8 Finite element mesh with boundary conditions

Analysis of results

The solution results for the finite element problem can be accessed in ANSYS via postprocessor.

Here we provide menu paths for the results that can be reviewed for structural analysis (after entering *Main Menu*).

В меню ANSYS это можно сделать, используя, например, следующие перемещения по меню:

- To display the deformed shape of the structure, select: General Postproc → Plot Results → Deformed shape
- In order to access the results for Degree of freedom solution (solution in terms of unknown variables), select: General Postproc → Plot Results → Contour Plot → Nodal Solu... → DOF Solution →
 - X-Component of displacement (distributions of the displacements U_x)
 - Y-Component of displacement (distributions of the displacements U_y)
 - Displacement vector sum (distribution of the displacement vector magnitude)
 - Electric potential (distribution of the electric potential)

To display the distribution of vector variables, select Vector Plot in General Postprocessor: General Postproc → Plot Results → Vector Plot → Predefined

- DOF solution → Translation U (distribution of the displacement vector)
- Flux & gradient → Elec field EF (distribution of the electric field vector)

The results of distribution of various components of stresses, strains, electric induction vector, etc. can be obtained in General Postprocessor in a similar way.

The program code for solving the example problem in ANSYS ends with the command **PLNSOL,U,Y** which shows the picture of displacement UY distribution (Fig. 11). We note that the example problem is axisymmetric in meridional section, where Oy -axis corresponds to Oz -axis in 3D, and Ox -axis corresponds to Or -axis. Therefore displacement UY is in fact the displacement u_z , and displacement UX is the displacement u_r and so on.

The main results are shown in Figs. 9-14. The distribution of mechanical and electric characteristics is shown for the undeformed shape of the disk.

```
DISPLACEMENT
STEP=1
SUB =1
TIME=1
DMX =.798E-09
```

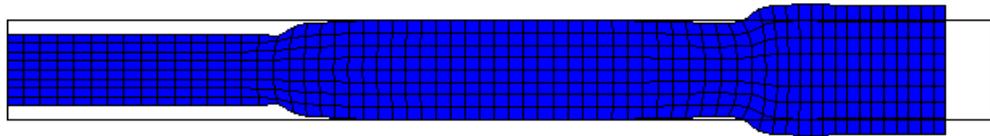


Fig. 9 Deformed shape of piezoelectric disk

```
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
UX (AVG)
RSYS=0
DMX =.798E-09
SMN =-.757E-09
SMX =.414E-09
```

ANSYS

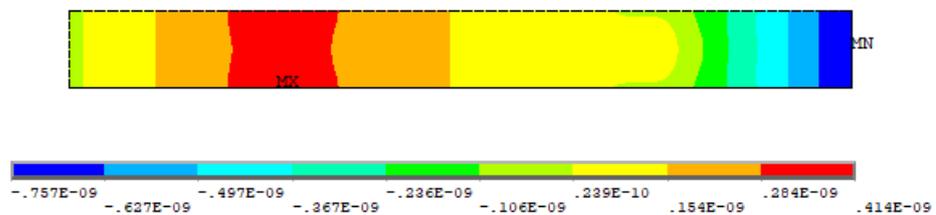


Fig. 10 Distribution of displacements UX

NODAL SOLUTION
 STEP=1
 SUB =1
 TIME=1
 UY (AVG)
 RSYS=0
 DMX =.798E-09
 SMN =-.262E-09
 SMX =.262E-09

ANSYS

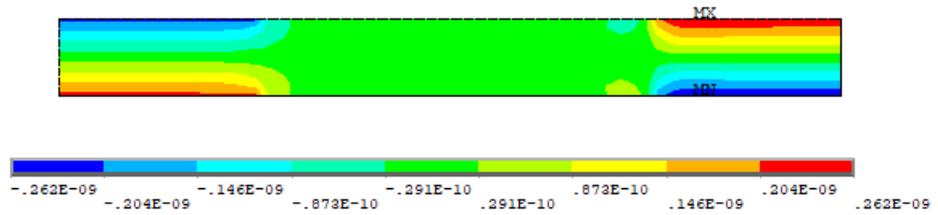


Fig. 11 Distribution of displacements UY

VECTOR
 STEP=1
 SUB =1
 TIME=1
 U
 NODE=1298
 MIN=0
 MAX=.798E-09

ANSYS

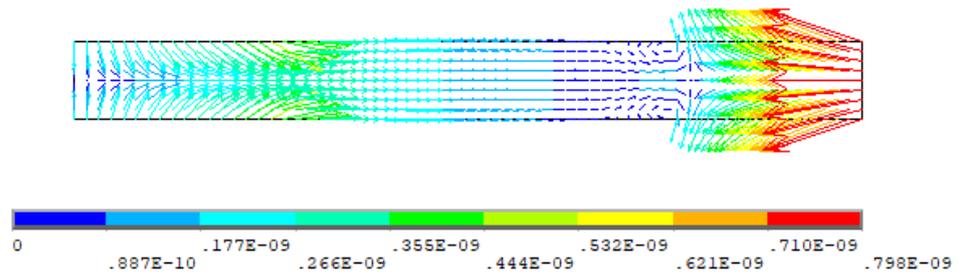


Fig. 12 Distribution of displacement vector U

NODAL SOLUTION
 STEP=1
 SUB =1
 TIME=1
 VOLT (AVG)
 RSYS=0
 DMX =.798E-09
 SMN =-1
 SMX =1

ANSYS

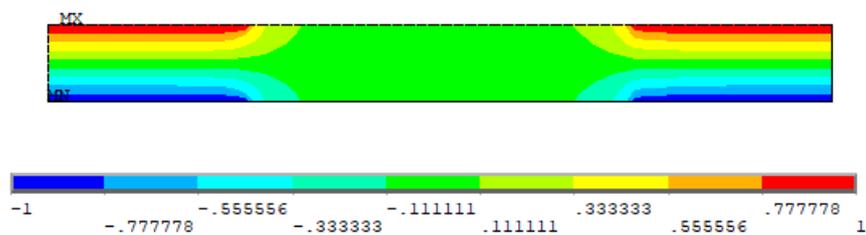


Fig. 13 Distribution of electric potential ϕ

```

VECTOR
STEP=1
SUB =1
TIME=1
EF
ELEM=352
MIN=38.2379
MAX=1433.76

```

ANSYS

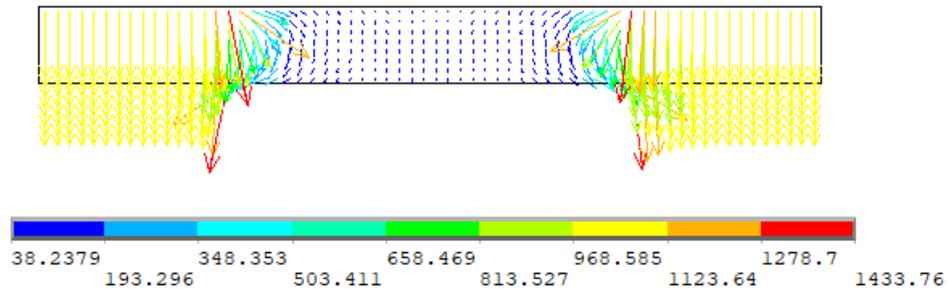


Fig. 14 Distribution of electric field vector \mathbf{E}

Let us analyse the obtained results from both physical and mathematical viewpoints. The difference of potentials $\Delta V = -2V_{inp}$ is applied between the electrodes 1 and 2 (i. e. between continuous electrodes in the zone $0 \leq X \leq R1$) and between the electrodes 3 and 4 (i. e. between ring electrodes in the zone $R2 \leq X \leq R$). Then we can expect that in the areas between these electrodes an electric field will be formed $E \sim \Delta V/H = -2V_{inp}/H = -1000$ (V/m), as $V_{inp} = 1$ (V), $H = 0.002$ (m). The same values of electric fields are indeed can be observed in Fig. 14 (yellow vectors). For the zone $0 \leq X \leq R1$ between continuous electrodes the direction of the polarization vector \mathbf{P} is opposite to the direction of the generated electric field \mathbf{E} . Therefore, due to the phenomenon of inverse piezoelectric effect, piezoelectric elements in this area should shrink. On the contrary, in the zone $R2 \leq X \leq R$ between ring electrodes the direction of the polarization vector \mathbf{P} coincides with the direction of the generated electric field \mathbf{E} . Then again, due to the phenomenon of the inverse piezoelectric effect, piezoelectric elements in the area $R2 \leq X \leq R$ should expand. Indeed, such deformations of the zones under the electrodes can be clearly seen in Fig. 9. Thus, the expected physics of electromechanical processes in the considered problem for a piezoceramic transducer with multielectrode coating is supported by the results presented in the figures above.

It is worth noting that finite element calculations are approximate. Therefore, it is necessary to address the question on the accuracy of the obtained results and determine the situations in which it is not worth to expect the convergence. In the above program code, the characteristics of the finite element mesh were defined in the following block of commands:

```

! Parameters of finite element mesh
SM=1.00 ! scaling multiplier
HDIV=8*SM ! Number of elements along the disk thickness
! (should be even number!)

```

R1DIV=16*SM ! Number of elements along the radial direction from 0 to R1
R12DIV=32*SM ! Number of elements along the radial direction from R1 to R2
R23DIV=16*SM ! Number of elements along the radial direction from R2 to R

Therefore, in order to analyze the convergence, we can perform computations with different values of the parameter SM . The results of this computation series are presented in the table below. As it can be seen from the table, the maximum values of the displacements get stabilized already at $SM = 0.75$ and continue stabilizing further with an increase of the parameter SM , i. e. when the sizes of finite elements become smaller the values of the displacements almost do not change. Thus, based on the data from the table, we can conclude that in order to estimate the maximal displacements for this problem it is enough to take $SM = 0.75$. However, already at $SM = 0.25$ the relative error in the determination of displacements is less than 3 %, which is enough for the majority of practical applications. The value $SM = 0.25$ means that we take two finite elements ($HDIV = 2$) along the disk thickness. This rather coarse mesh is satisfactory to define the displacements, as the considered problem is static and has rather simple geometry, and the elements **PLANE223** are finite elements with quadratic approximation for each canonic variable.

Table. Data to analyze the convergence of results

SM	0.25	0.50	0.75	1.00	1.25	1.50	1.75	2.00
$\max u_y \cdot 10^{-12}$ (M)	255	260	262	262	262	262	262	262
$\max E $ (B/M)	1001	1141	262	1293	1564	1685	1798	1904

Meanwhile, as it can be seen from Fig. 14, the electric field vector changes significantly in the vicinity of the boundaries of the electrode surfaces (red vectors). Data from the table shows that, when the mesh density increases, the maximal values of the electric field vector magnitude also increase, and we observe no convergence for these values. This effect can be expected, as in the electroelasticity problems the surface electrodes are the concentrators for various components of the electromechanical fields of stresses, strains, electric field intensity and electric induction (i. e. for gradient or flow values defined by the derivatives of displacements and electric potential). The role of surface electrodes in these problems are similar to the roles of the stamps with non-smooth boundaries in contact problems. Therefore, as in similar contact problems, in the vicinity of the electrode boundaries some components of stresses, strains, electric field intensity and electric induction can have root singularities, such as $r^{1/2}$, where r is the distance from the electrode boundary. Naturally, when using conventional isoparametric finite elements, we will observe an increase of the values of these fields in the vicinity of the electrode boundaries. In practice, it means that specific values of such flow variables in a small area near the electrode boundaries do not have much sense,

but far away from the electrode boundaries, the convergence will take place. Their integral values (for example, the total electric charge) will also converge.

The idea stated above can be illustrated by a graph (Fig. 15) that shows the change of the axial component of the electric field intensity E_z (EF, Y in ANSYS) along the radius of the disk on its upper surface ($0 \leq X \leq R$, $Y = H/2$ in ANSYS). This graph can be plotted in ANSYS General Postprocessor by the following commands:

```
! Postprocessor plot of the graphs
/PLOPTS,INFO,ON
/COLOR,CURVE,WHIT,1,6 ! color for the graphs - white
! (will be black after inversion)
/COLOR,GRID,WHIT, ! color for mesh - white
! (will be black after inversion)
/PLOPTS,TITLE,OFF ! no title for the graph
/PLOPT,FRAME,OFF ! no frame
/AXLAB,X,r label for the X-axis (r-axis)
/AXLAB,Y, E_Z ! label for the Y-axis (z-axis)
/GROPT,DIVX,4 ! Number of divisions along X-axis
/GROPT,DIVY,4 ! Number of divisions along Y-axis
/GROPT,DIG2,1 ! Number of digits after decimal point

! Commands to define a path by a number of points
PATH,XX,4,,120
PPATH,1,,0,H/2
PPATH,2,,R1,H/2
PPATH,3,,R2,H/2
PPATH,4,,R,H/2 ! Define a variable E_Z for plotting a graph along the path
PDEF,E_Z,EF,Y ! Define other variables (if needed)
PDEF,T_ZZ,S,Y $ PDEF,T_RR,S,X
PDEF,D_Z,D,Y $ PDEF,D_R,D,X $ PDEF,E_R,EF,X
PLPATH,E_Z ! Plot a graph of E_Z along the path
```

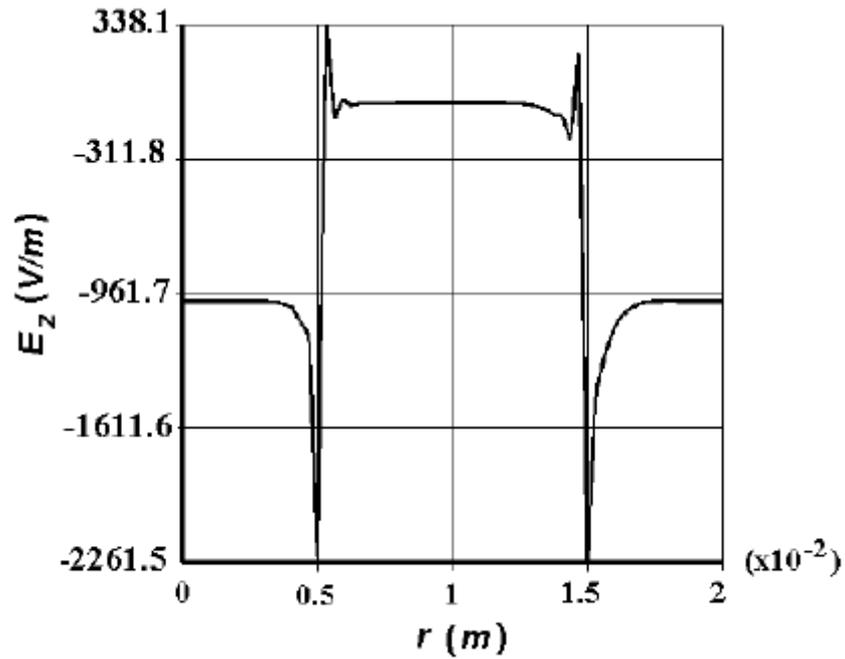


Fig. 15 Behavior of the E_z -component of the electric field vector on the disk surface

The finite element results presented in Fig. 15 for the dependence $E_z = E_z(r)$ are not accurate in the vicinity of the electrode $R_1 = 0.5 \cdot 10^{-2}$ (m) and $R_2 = 1.5 \cdot 10^{-2}$ (m). If, for example, at $r < R_1$ this curve is somewhat similar to a function with a root singularity $k\sqrt{R_1^2 - r^2}$, then a jump of the function at $r > R_1$ is completely determined by the error of finite element approximations and the averaging techniques adopted here for calculation of the field gradients. However, far away from the electrode boundaries the field is determined with sufficient accuracy.

By analogy we can analyze the behavior of other gradient values of electromechanical fields: E_r , D_z , D_r , σ_{zz} , etc.