Task №1
PLANE STRESS TENSION OF A PLATE WITH A HOLE

KEYWORDS
1. Linear theory of elasticity
2. Static analysis, structural analysis
3. Plane problem (plane stress)
4. Stress concentration

PROBLEM DISCRIPTION

A thin rectangular plate with the length of 2*a*; *a*=5 (cm) and the width of 2*b*; *b*=2 (cm) has a hole in the center with the radius *R*=0.25 (cм) (Fig. 1). The plate is made of an elastic isotropic material with the Young's modulus $E=2\cdot10^6$ (kgf/cm^2) and the Poisson's ratio ν=0,3. The plate is being stretched by the distributed load $p=0{,}1\cdot10^6$ (kgf/cm^2), applied to its left and right edges. The objective of the problem is to perform plane stress structural analysis and define maximal stresses in the plate.

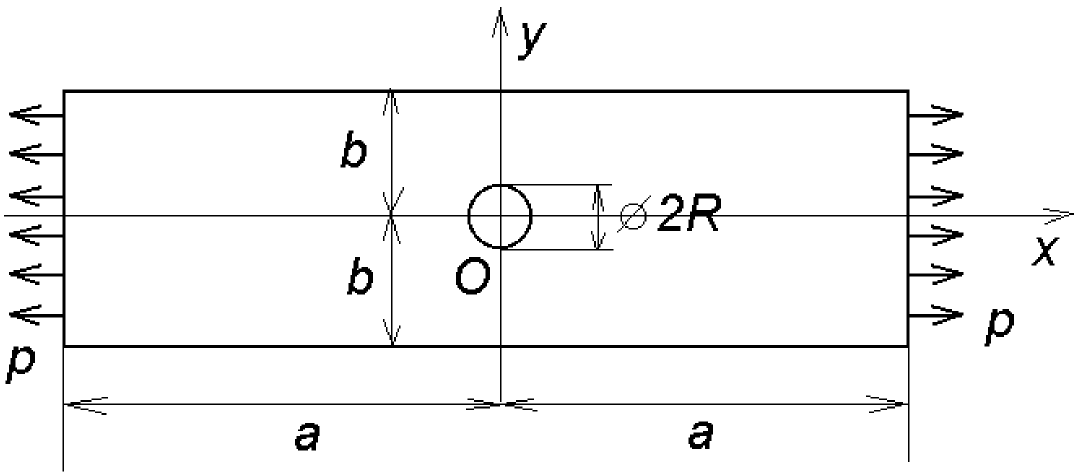


Figure 1. Scheme of a plate with a hole with boundary conditions

INTRODUCTORY NOTES

It is necessary to note that the user should control the consistency of the system of units for the input values. Here the chosen system of units is cm for measuring length and kg for measuring mass. So the pressure load, Young's modulus and stresses are measured in kgf/cm^2 (kilogram force per square centimeters), where 1 kgf/cm^2 = 98066.5 Pa.

In this problem, the hole introduces a perturbation into a uniform stress state of the plate loaded in uniaxial direction. In the vicinity of the hole there is an increase of the stresses known as the stress concentration. An analogous problem for an infinite plate stretched by distributed loads at infinity is called a Kirsch problem, and the solution for such problem can be obtained analytically. The Kirsch problem is the fundamental problem of the elasticity theory on the stress concentration. In the Kirsch problem the maximal stresses arise in the point (0, *R*) and are equal to *3p*. These stresses are tangential stresses.

In the example problem the stress, strain and displacement fields are inherently inhomogeneous around the hole, therefore for accurate computations it is necessary to condense finite element mesh around the hole.

THEORETICAL BACKGFROUND

In an assumption of a plane stress state the displacements of the plate in the region Ω, in the *xy*-plane are characterized by the displacement vector $\underline{U}=\{U_x, U_y\}=\{U, V\}$, where $U=U(x, y)$, $V=V(x, y)$.

The components ε_{xx}, $\varepsilon_{xy} = \varepsilon_{yx}$, ε_{yy} of the strain tensor $\underline{\underline{\varepsilon}} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix}$, are related to the components of the displacement vector $\underline{U}$ by formulas:

$$S_{xx} = \varepsilon_{xx} = \partial U / \partial x \,;\; S_{yy} = \varepsilon_{yy} = \partial V / \partial y \,; \tag{1}$$

$$S_{xy} = \varepsilon_{xy} = (\partial U / \partial y + \partial V / \partial x)/2$$

The constitutive relations between mechanical stresses and strains in an elastic isotropic medium under plane stress state have the form

$$\begin{aligned} T_{xx} &= \sigma_{xx} = \lambda^*(S_{xx} + S_{yy}) + 2\mu S_{xx} \\ T_{yy} &= \sigma_{yy} = \lambda^*(S_{xx} + S_{yy}) + 2\mu S_{yy} \\ T_{xy} &= \sigma_{xy} = 2\mu S_{xy} \end{aligned} \tag{2}$$

where

$$\lambda^* = \frac{2\lambda\mu}{\lambda + 2\mu} \tag{3}$$

$$\lambda = \frac{\nu E}{(1+\nu)(1-2\nu)}, \quad \mu = \frac{E}{2(1+\nu)} \tag{4}$$

$\underline{\underline{\sigma}} = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy} \end{pmatrix}$ is the stress tensor, σ_{xx}, $\sigma_{xy} = \sigma_{yx}$, σ_{yy} are the components of the stress tensor.

The coefficients λ and μ from (4) are known as Lame's coefficients. Often the coefficient μ is denoted by G and has the meaning of the shear modulus. The module E from (4) is called the Young's modulus, and ν is called the Poisson's ratio.

The equilibrium equations for an elastic medium in a plane problem have the form

$$\partial T_{xx} / \partial x + \partial T_{xy} / \partial y = 0 \tag{5}$$

$$\partial T_{xy} / \partial x + \partial T_{yy} / \partial y = 0 \tag{6}$$

Substituting (2) and (1) into (5), (6) gives an elliptic system of partial differential equations of the second order for unknown functions of displacements U and V.

This system should be supplemented by the boundary conditions on the boundary $\Gamma = \partial\Omega$. Together with boundary conditions, this system constitutes a *boundary-value problem*.

Let the boundary Γ be divided into two subsets Γ_u and Γ_σ. At the part of the boundary Γ_u the components of the displacement vector are considered to be known:

$$U = U, \qquad V = V, \qquad \{x,y\} \in \Gamma_u \tag{7}$$

At the part of the boundary Γ_σ the pressure (stress vector) $p = \{p_x, p_y\}$ is defined

$$T_{xx}n_x + T_{xy}n_y = p_x \qquad T_{xy}n_x + T_{yy}n_y = p_y, \quad \{x,y\} \in \Gamma_\sigma \tag{8}$$

where $n = \{n_x, n_y\}$ is the outward unit normal vector to the boundary Γ.

In elasticity theory, there are two main types of boundary conditions. In notation of the finite element method these two types of boundary conditions are known as *essential* and *natural*. Essential boundary conditions are the conditions that are imposed explicitly on the unknown function (a primary variable); they correspond to Dirichlet boundary conditions in a boundary value problem. Natural boundary conditions are given in terms of the derivatives of unknown functions (secondary variables, for example, stresses in linear elasticity), they correspond to Neumann boundary conditions. Natural boundary conditions will be satisfied automatically after the problem is solved.

Boundary conditions (7) in terms of displacements are essential boundary conditions, also known as Dirichlet boundary conditions, or boundary condition of the first kind. Values $U = 0$, $V = 0$ in (7) usually correspond to a rigidly fixed part of the boundary Γ_u.

Boundary conditions (8) in terms of stresses are natural boundary conditions, also known as Neumann boundary conditions, or boundary condition of the second kind. When $p_x = 0$, $p_y = 0$, the part of the boundary Γ_σ is considered to be a free boundary. As a vector-function of *x*, *y*, the stress vector $p = \{p_x, p_y\}$ can include concentrated force vectors $F = \{F_x, F_y\}$.

USING ANSYS TO SOLVE THE PROBLEM

The problem can be simulated and solved in ANSYS using either interactive mode, or command mode, or a combination of both. An interactive mode of solving the problem step by step in GUI is described in file St2LS_1(ANSYS).doc. File St2LS_1.inp contains input listing of commands in ANSYS APDL (ANSYS Parametric Design Language). This file can be executed in ANSYS from menu File → Read Input from… After that, the results can be viewed in General Postprocessor in an interactive mode.

An example similar to the considered problem is included in ANSYS Verification Manual, see VM142 for details.

USING FLEXPDE TO SOLVE THE PROBLEM

Input file for solving the problem in FlexPDE is called St2LS_1.pde.

REVIEWING AND ANALYZING RESULTS

Let us review results obtained in ANSYS.

Due to the symmetry of the problem, including both geometrical symmetry of the domain and symmetrical boundary conditions, we can consider a quarter of the plate. Fig. 2 illustrates the resulting area A3 with the keypoint and area numbers. (Menu path: Plot->Areas, for showing numbers of the entities go to PlotCtrls->Numbering->tick Area numbers, Keypoint numbers).

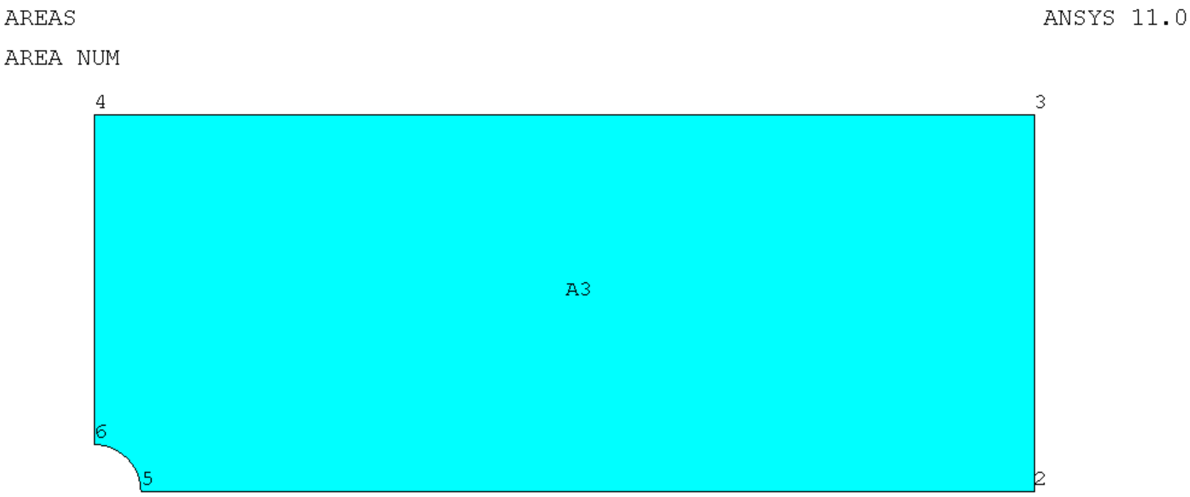
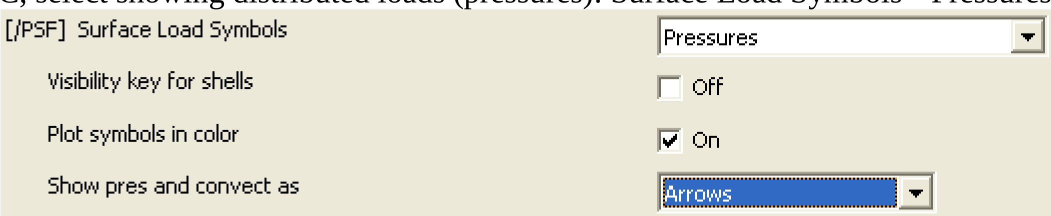


Figure 2. Problem domain (area with keypoint numbers)

The finite element model of the problem with applied boundary conditions is shown in Fig. 3 (Menu path: Plot->Elements, for showing boundary conditions go to PltCtrls->Symbols->tick All applied BC, select showing distributed loads (pressures): Surface Load Symbols->Pressures)



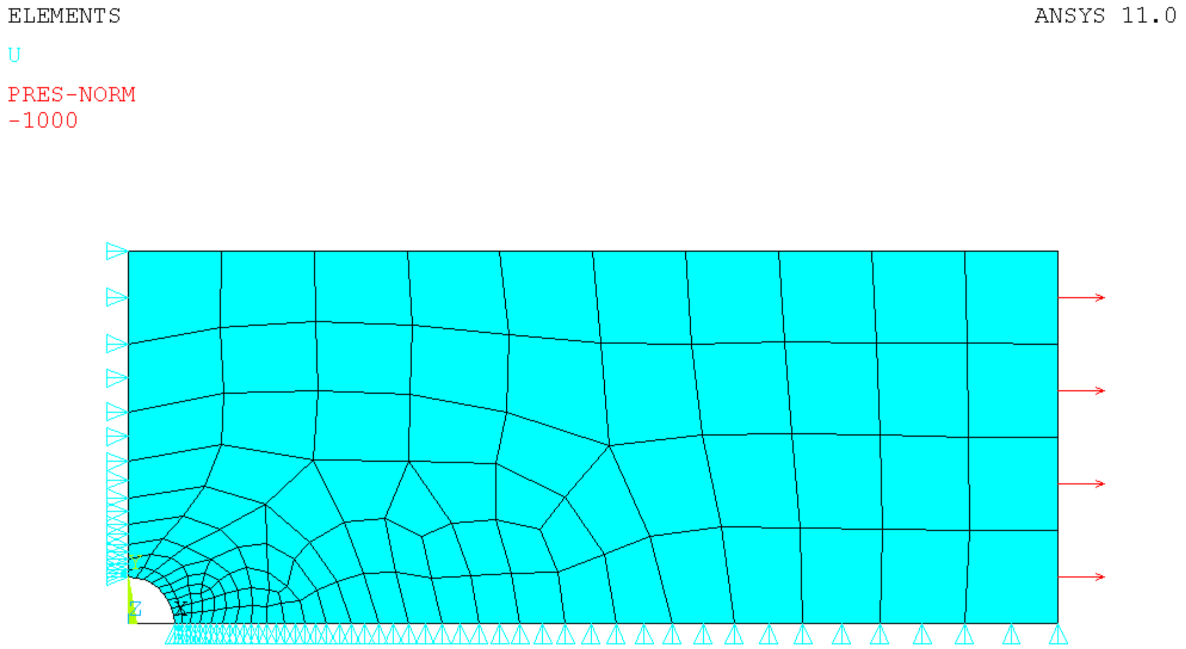


Figure 3. Finite element mesh with boundary conditions

The area is meshed with quadrilateral eight-node finite elements PLANE82 suitable for 2D structural analysis. The finite element PLANE82 has two degrees of freedom (Ux and Uy) in each node.

Some results of the computations can be seen in Fig. 4-6. Fig. 4 illustrates the distribution of the displacements U_x (Menu path: General Postproc → Plot Results → Contour Plot → Nodal Solu → DOF Solution → X-Component of displacement).

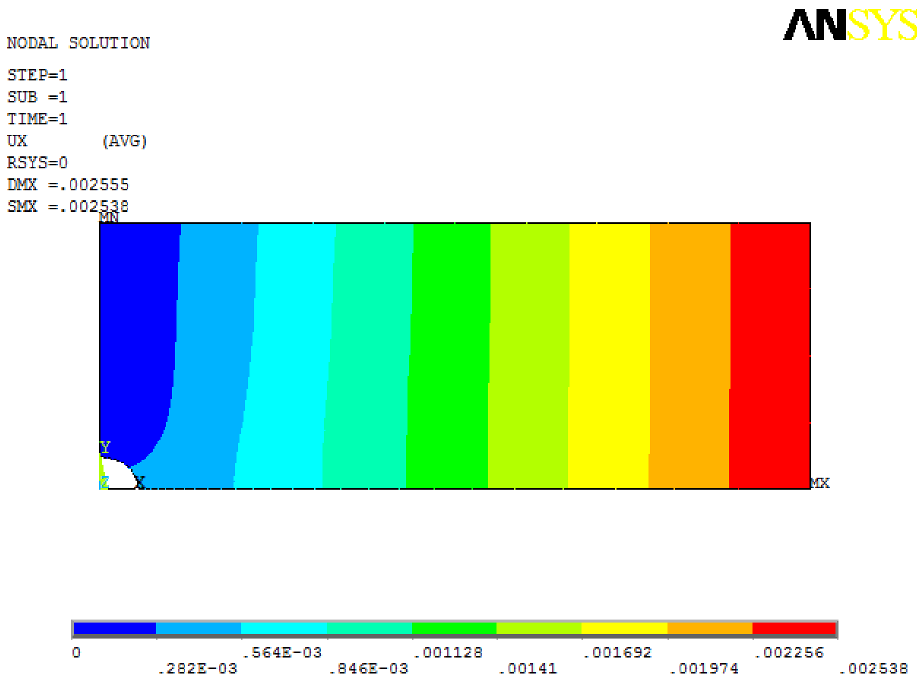


Figure 4. Distribution of the displacements U_x

Fig. 5 shows the distribution of the axial stresses T_{yy} (Menu path: General Postproc → Plot Results → Contour Plot → Nodal Solu → Stess →Y-Component of stress).

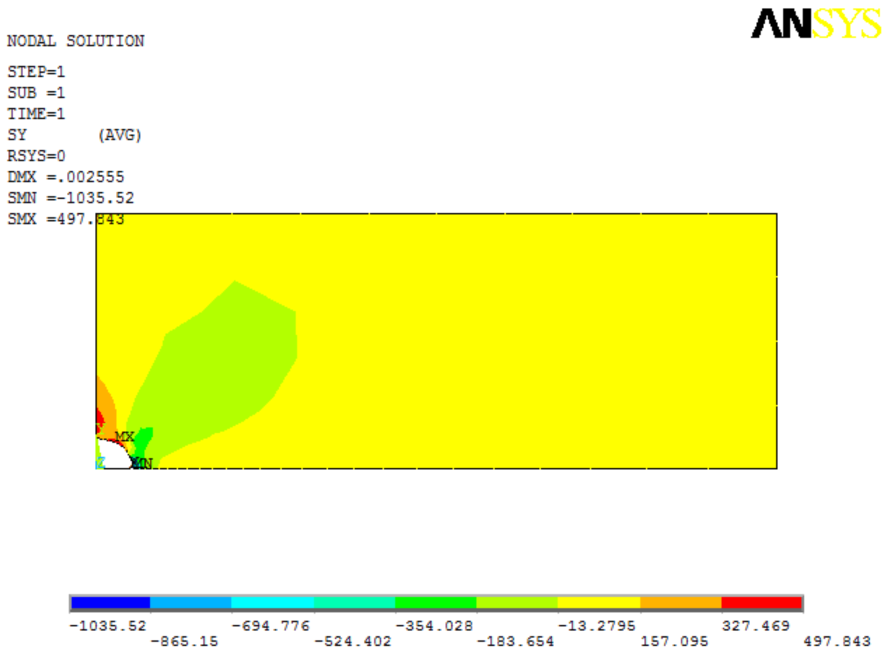


Figure 5. Distribution of axial stresses T_{yy}

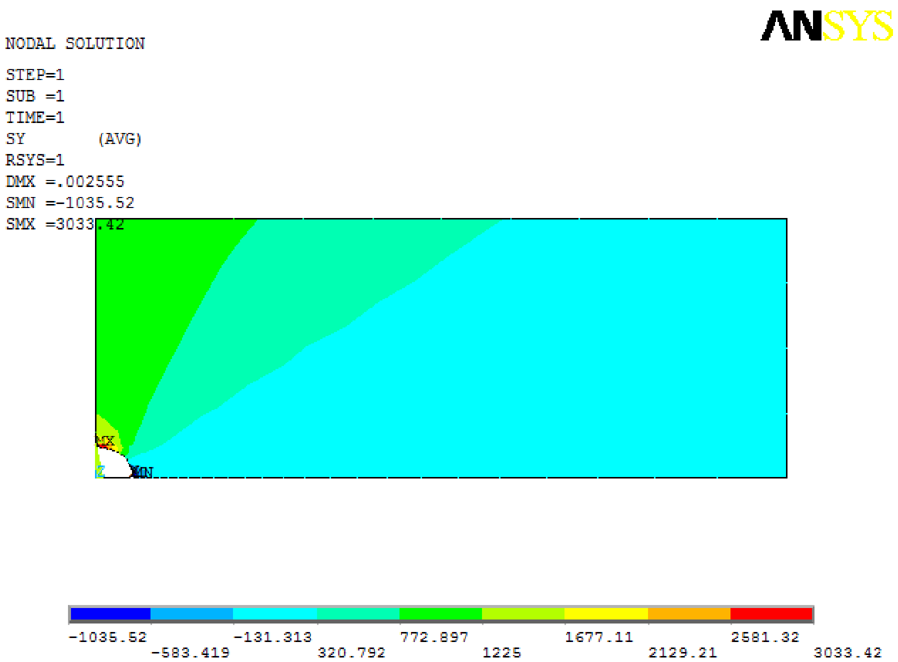


Figure 6. Distribution of tangential stresses $T_{\theta\theta}$

Fig. 6 shows the distribution of the tangential stresses $T_{\theta\theta}$ (Menu path: General Postproc → Options for Outp → Results coordinate system → Global Cylindrical; Plot Results → Contour Plot → Nodal Solu → Stess →Y-Component of stress (y corresponds to θ in Cylindrical coordinate system).

As it can be seen from Fig. 6, maximal stresses are concentrated around the hole.