Laboratory work №1
PLANE STRESS TENSION OF A PLATE WITH A HOLE
Test problem

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 (см) (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 $p=10^3$ (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.

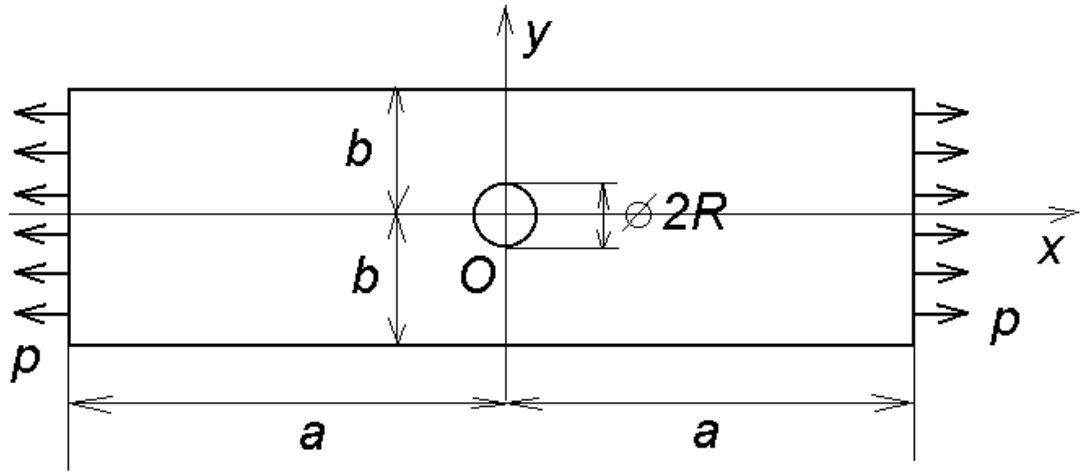


Figure 1. Scheme of a plate with a hole with boundary conditions

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:

$$S_{xx}=\varepsilon_{xx}=\partial U/\partial x;\ S_{yy}=\varepsilon_{yy}=\partial V/\partial y; \qquad (1)$$
$$S_{xy}=\varepsilon_{xy}=(\partial U/\partial y+\partial V/\partial x)/2$$

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

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

where

$$\lambda^{*}=\frac{2\lambda\mu}{\lambda+2\mu} \qquad (3)$$

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

$\underline{\underline{\sigma}}=\begin{pmatrix}\sigma_{xx} & \sigma_{xy}\\ \sigma_{yx} & \sigma_{yy}\end{pmatrix}$ 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

$$\partial T_{xx}/\partial x+\partial T_{xy}/\partial y=0 \qquad (5)$$

$$\partial T_{xy}/\partial x+\partial T_{yy}/\partial y=0 \qquad (6)$$

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:

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

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

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

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 FLEXPDE TO SOLVE THE PROBLEM

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

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(GUI 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.

To solve the problem about the plate tension, we need to use the finite elements for Structural analysis. Such elements will have degrees of freedom (DOF) UX, UY, UZ, which are displacements in the nodes. The finite elements for 3D simulation are called Solid. As the plate has small thickness compared to its width and length, and all the loads are applied to its lateral edges, we can consider 2D model of the plate and simulate plane stress behavior (see Fig. 2). For two-dimensional case we can use plane or shell finite elements (which have thickness).

2D finite elements can have the shapes of triangle or quadrilateral. 3D finite elements have broader range of shapes, including tetrahedron, prism, hexahedron. Fig. 3 shows linear elements (with nodes only in the element vertices) and quadratic elements (with internal nodes on the element edges).

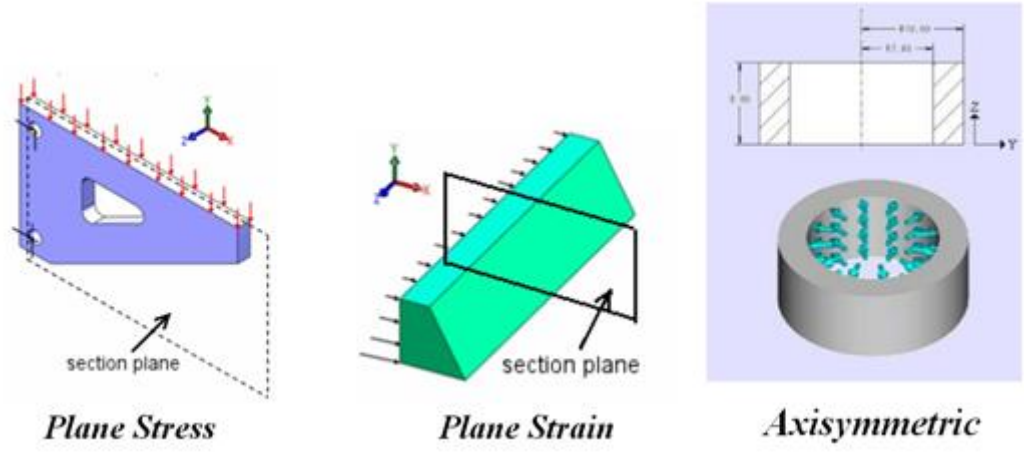


Figure. 2 2D problem setting

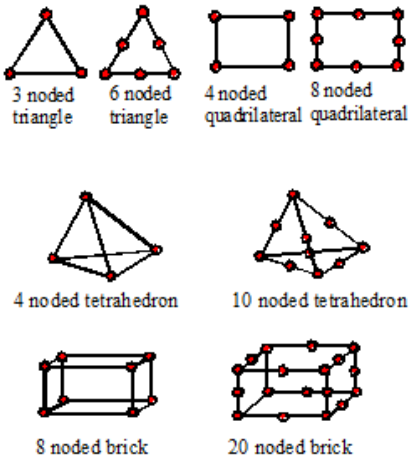
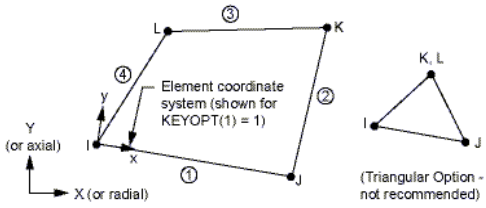
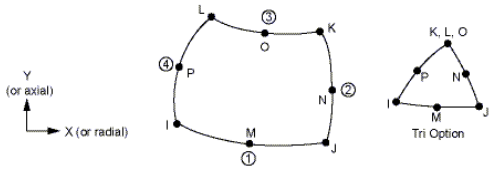
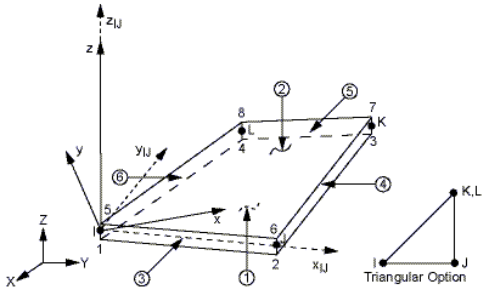
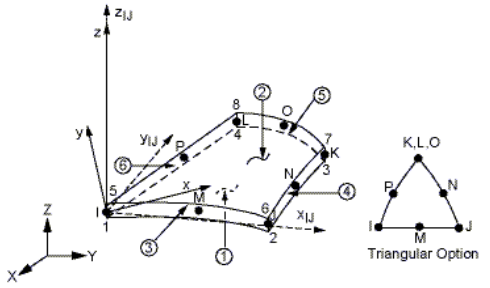
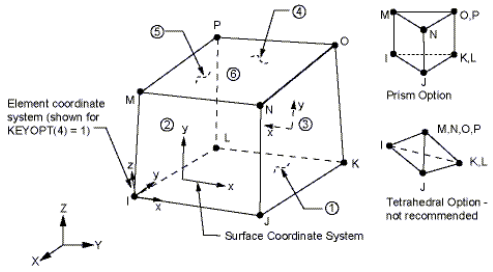
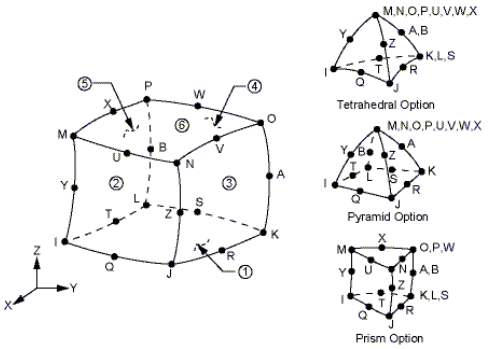
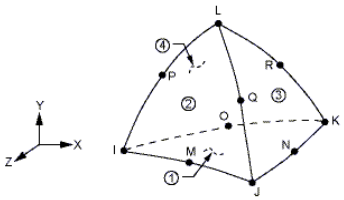


Figure. 3. Types of linear and quadratic 2D and 3D finite elements

The table below contains the elements which can be used for linear structural analysis.

Table 1. Elements available for linear structural analysis in Ansys (old and new versions)

Element type	**Element name**	**Element shape, number of nodes, oder of approximation**	**Degrees of freedom and element behavior**
2D plane finite elements	**PLANE42, 182** 2-D Structural Solid	 4 node linear quadrilateral	**UX, UY** **KEYOPT(3)** Element behavior: **0 -** Plane stress **1 -** Axisymmetric **2 -** Plane strain **3 -** Plane stress with thickness input
	PLANE82, 183 2-D 8-Node Structural Solid	 8 node quadratic quadrilateral	**UX, UY** **KEYOPT(3)** Element behavior: **0 -** Plane stress **1 -** Axisymmetric **2 -** Plane strain **3 -** Plane stress with thickness input
Shell finite elements	**SHELL63, 181** Elastic Shell	 4 node linear quadrilateral shell	**UX, UY, UZ, ROTX, ROTY, ROTZ** KEYOPT(1) Element stiffness: 0 -- Bending and membrane stiffness 1 -- Membrane stiffness only 2 -- Bending stiffness only

	SHELL93, 281 8-Node Structural Shell	 8 node quadratic quadrilateral shell	**UX, UY, UZ, ROTX, ROTY, ROTZ**
3D solid finite elements	**SOLID45, 185** 3-D Structural Solid	 8 node linear hexahedron	**UX, UY, UZ**
	SOLID95, 186 3-D 20-Node Structural Solid	 20 node quadratic hexahedron	**UX, UY, UZ**
	SOLID92, 187 3-D 10-Node Tetrahedral Structural Solid	 20 node quadratic hexahedron	**UX, UY, UZ**

Solid and finite element model

Note that the plate geometry and boundary condition are symmetric with respect to the central axes of the plate (see Fig. 1). Therefore, we can consider a quarter of the plate and set symmetry boundary conditions on the respective lines or edges of symmetry.

The following fragment of the file St2LS_1.inp shows how to solve the **plane stress** problem of the plate tension APDL ANSYS. Here all necessary parameters are set first, and then the materialproperties are defined. The finite element type for is an 8-node quadrilateral quadratic element **PLANE82** (with the nodes on the element edges).

The solid model of the plate is built using keypoints to describe the main domain, from which the circular are is subtracted. **AMESH** commend builds the finite element mesh of the resulting area.

```
/TITLE, Plane Stress tension of an elastic plate with a hole
/PREP7
! Due to the symmetry of the problem we consider a quarter of the plate
A=5   ! length of the plate quarter
B=2   ! width of the plate quarter
```

```
R=0.25      ! radius of the hole
H=0.1       ! thickness of the plate
P=1e3       ! magnitude of tensile load(kG/cm^2)
!1kG = 9.8 N/m^2
MP,EX,1,2e6       ! Young's moduus EX=2*10+6  (kG/cm^2)
MP,NUXY,1,0.3     ! Poisson's ratio NUXY=0.3

ET,1,PLANE183 ! Eight-node finite element PLANE183 (plane stress)
! ***************************************************
! For stress analysis of the plate we can also choose shell element SHELL63, in
this case uncomment the following commands
!ET,1,SHELL63
!R,1,H
!P=P*H         ! pressure for a length unit (kG/cm)
! ***************************************************

K,1,0,0     ! Keypoints of the plate boundary: point number and coordinates
K,2,A,0
K,3,A,B
K,4,0,B
A,1,2,3,4   ! Define area 1 using four keypoints
! Command A defines an area by connecting keypoints (max 18 points). Keypoints
must be input in a clockwise or counterclockwise order around the area.
APLOT,1     ! Show area 1

PCIRC,R     ! Define area 2 - a cirlce with radius R and center in (0,0)

ASBA,1,2    ! Substract area 2 from area 1
APLOT,ALL   ! Show resulting area 3

! Define parameters for finite element mesh
! KESIZE Specifies the edge lengths of the elements nearest a keypoint
KESIZE,ALL,B/4
KESIZE,5,R/6 ! set element edge length near keypoint 5
KESIZE,6,R/6 ! set element edge length near keypoint 6

AMESH,ALL         ! mesh area all areas (area 3)
FINISH
```

Fig. 4 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).

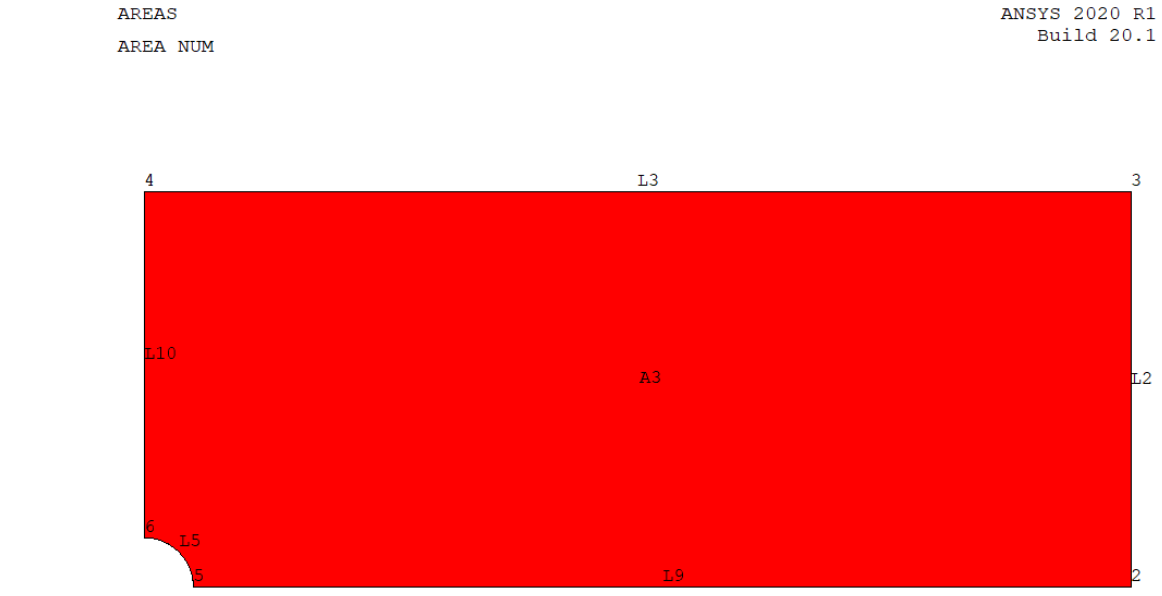


Figure 4. Problem domain (area with keypoint and line numbers)

Fig. 5 shows the lines which constitute the resulting are with the line and keypoint numbers. (Menu path: Plot->Lines, for showing numbers of the entities go to PlotCtrls->Numbering->tick Line numbers, Keypoint numbers).

Fig. 6 shows the resulting finite element mesh with elements and their nodes показаны полученные в результате конечно-элементного разбиения элементы и узлы. . (Menu path: Plot->Elements, Plot->Nodes).

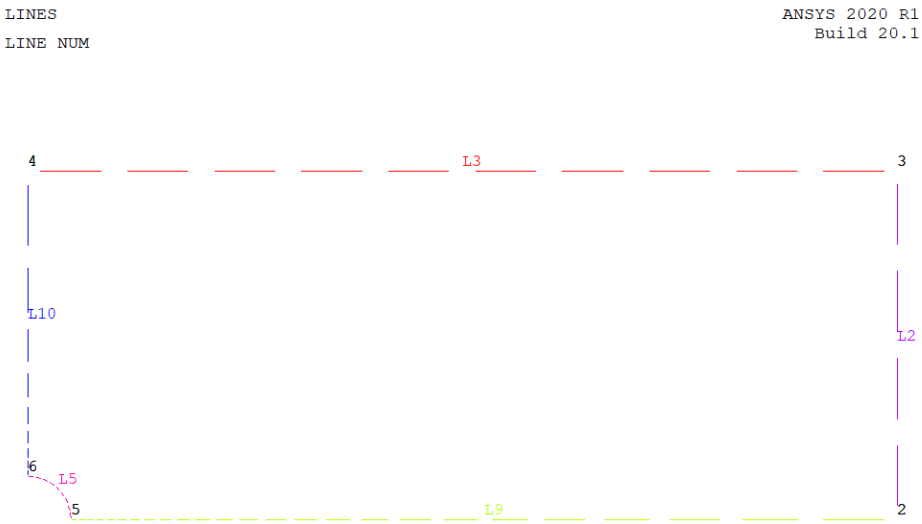


Figure 5. Lines with their numbers and keypoint numbers

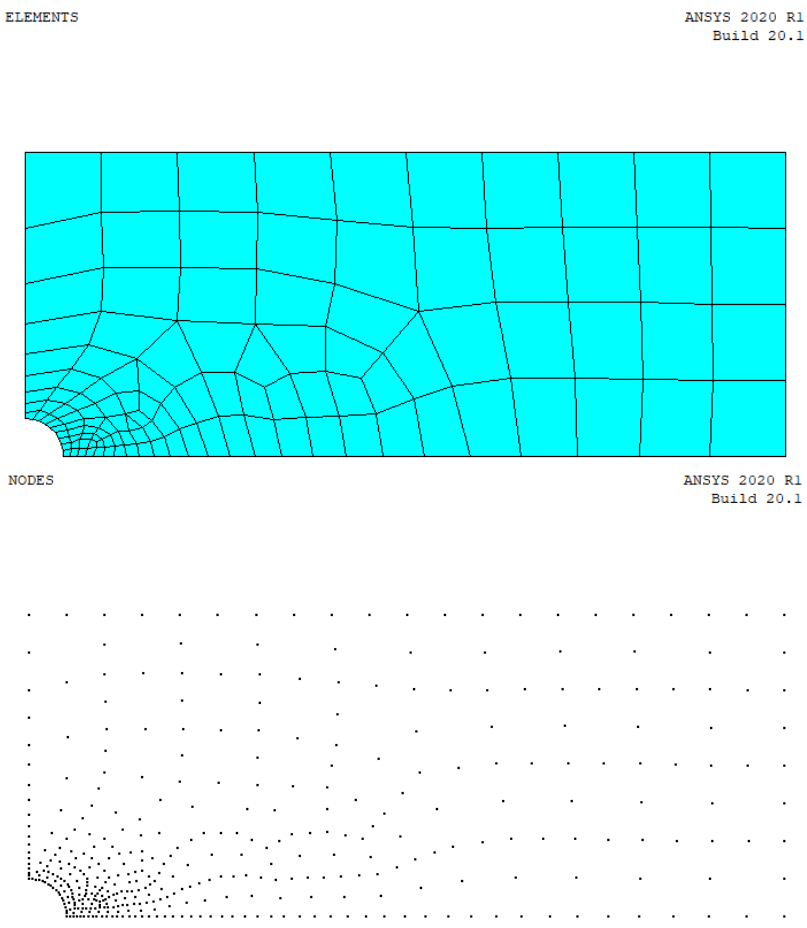


Figure 6. Elements PLANE183 and their nodes

Setting boundary conditions

The plate is being stretched by its lateral edges by the distributed load (pressure) $p=1\cdot10^3$ ($кГ/см^2$), applied to its left and right edges. The remaining part of the boundary is free from stress. We simulate only one quarter of the plate in 2D case, therefore we need to put symmetry boundary conditions on the lines of symmetry (the lines where we mentally cut the plate) and the load on the right edge of the plate quarter. The rest of the boundary is assumed to be free, and this is the default boundary condition, which is not necessary to specify.

The boundary conditions can be set either for finite element model entities (nodes and elements) or for solid model entities (keypoints, lines and areas). At the stage of solving the problem all boundary conditions set on solid entities are transferred to finite element model. The boundary conditiones set on solid entities have priorities over the boundary conditions set on finite element entities. If the user specifies boundary conditions both on solid and finite element entities, a warning will be shown calling user's attention to the possibility of overwriting finite element boundary conditions by solid ones set at the same place.

Let us consider the code fragment from the file St2LS_1.inp. Here the distributed load (pressure) is set on nodes (finite element model entities), and both symmetry conditions are set on lines (solid model entities). The commented commands enable us to set the same symmetry conditions directly on the nodes.

In the example problem, the right edge of the plate lies on the line x=a, therefore the stretching surface load is set on the nodes with coordinates x=a with negative sign for pressure: PRES = -P (for a compressive load, we set positive pressure with a positive sign: PRES = P). For our example problem (see Fig. 1) the symmetry boundary conditions should be set on the left edge of the plate (which lies on the line x=0, axis OY is vertical) and on the bottom edge of the plate (which lies on the line y=0, axis OX is horizontal). The symmetry condition on the vertical axis OY (x=0) indicates no horizonal displacements: UX=0. Similarly, the symmetry condition on the horizontal axis OX (y=0) indicates no vertical displacements: UY=0.

```
/SOLU
ANTYPE,STAT           ! set analysis type: static
NSEL,S,LOC,X,A   ! select all nodes with coordinate X=A
SF,ALL,PRES,-P   ! for all selected nodes set surface load PRES = -P
NSEL,ALL         ! select all nodes
DL,9,,SYMM       ! symmetry condition on line 9 (all lines with Y=0)
DL,10,,SYMM      ! symmetry condition on line 10 (all lines with X=0)

! *****************************************************
! For finite element with degrees of freedom UX, UY the previous symmetry
conditions on lines 9 and 10 are equivalent to the following commands
!NSEL,S,LOC,X,0 !Select all nodes with coordinate x=0
!D,ALL,UX,0 ! for all selected nodes set displacement ux=0
!NSEL,S,LOC,Y,0 !Select all nodes with coordinate y=0
!D,ALL,UY,0 ! for all selected nodes set displacement uy=0
!NSEL,ALL ! select all nodes
! *****************************************************

SOLVE            ! Solve finite element system of equations
FINISH
```

D and DL commands set constraints on the degrees of freedom (DOF). D command sets constraints on the nodes, and DL command sets constraints on the lines. SF and SFL commands set surface load. SF command sets surface load on the nodes, and SFL command sets surface load on the lines.

Note that the nodes can be selected both by coordinates and by lines, on which the nodes are located. For example, in the test problem the right edge is the line L2 (see Fig. 3). Hence, the surface load can be set in the following way:

```
LSEL,S,LINE,,2   ! Select line 2
NSLL,S,1 ! Select the nodes located on the selected line
! 2nd argument is the key to select the nodes: 1 – internal and key points
(line ends); 0 – select only nodes interior to selected line
SF,ALL,PRES,-P   ! Set load pressure for all selected nodes
```

The resulting finite element model of the problem with applied boundary conditions is shown in Fig. 5 (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)

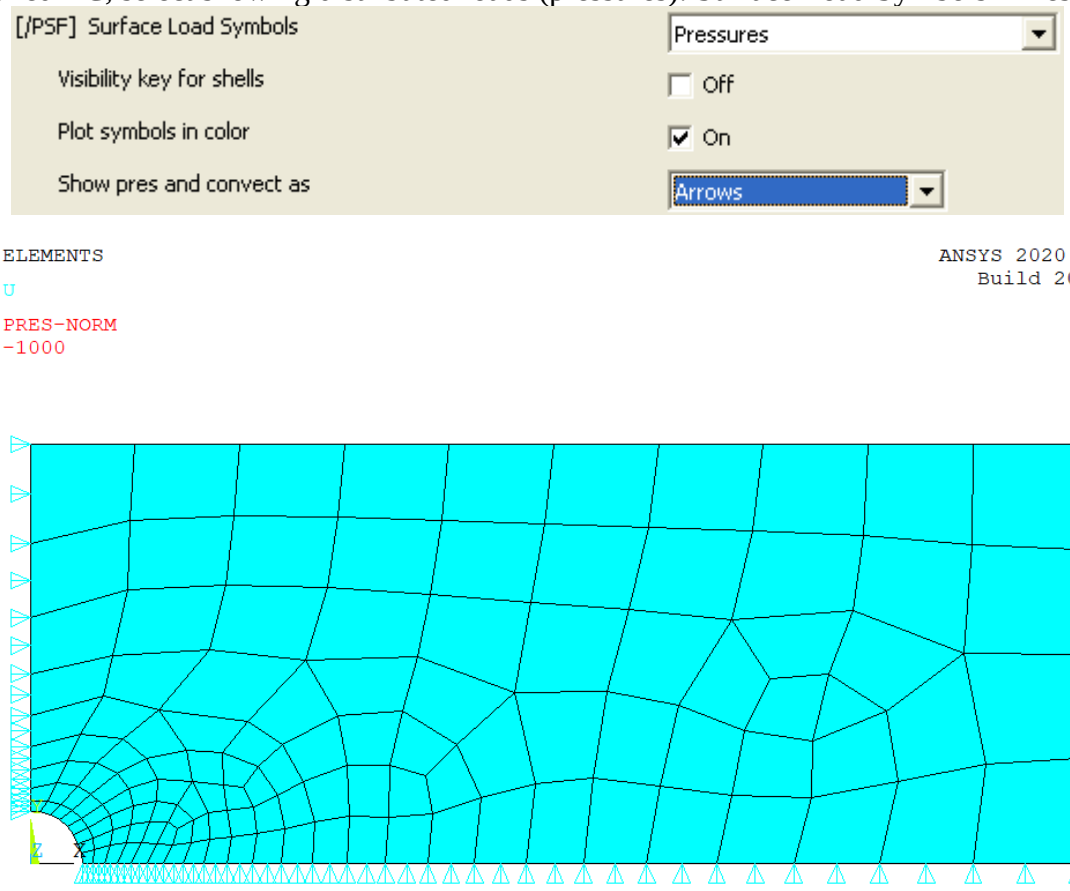


Figure 7. Finite element mesh with boundary conditions

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.

REVIEWING AND ANALYZING RESULTS

Let us plot several computation results. For instance, we can start with the deformed shape (Fig. 8). Menu path: General Postproc → Plot Results → Deformed shape → Def + undef edge.

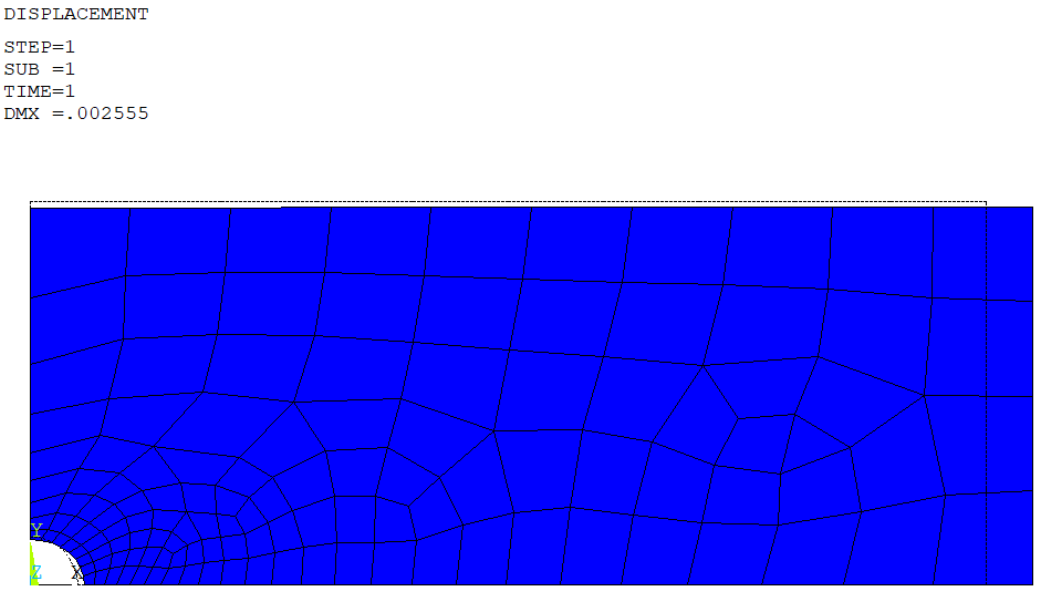


Fig. 8. Deformed mesh with undeformed edge

Then we can plot the distribution of the displacements U_x (Fig. 9), the distribution of axial stresses T_{yy} (Fig. 10) and the distribution of tangential stresse $T_{\theta\theta}$ (Fig. 11).

Menu path for plotting the picture of U_x distribution: General Postproc → Plot Results → Contour Plot → Nodal Solu → DOF Solution → X-Component of displacement.

Important note: in all modern versions of ANSYS the display of the results for the solution derivatives, such as stresses and stresses, may not be available Plot Results immediately. In this case when working in interactive mode one can open Results Viewer first or read the results of the last set General Postproc → Read Results → Last Set. In a command moder or for an input file, just add a command to read the results of the last set to get access to these results: **SET, LAST**

Menu path for plotting the picture of T_{yy} distribution: General Postproc → Plot Results → Contour Plot → Nodal Solu → Stress → Y-Component of stress.

Menu path for plotting the picture of $T_{\theta\theta}$ distribution: General Postproc → Options for Outp → Results coordinate system → Global Cylindrical; Plot Results → Contour Plot → Nodal Solu → Stess →Y-Component of stress (when we set results coordinate system to global cylindrical, the y-axis will correspond to coordinate θ in global cylindrical system).

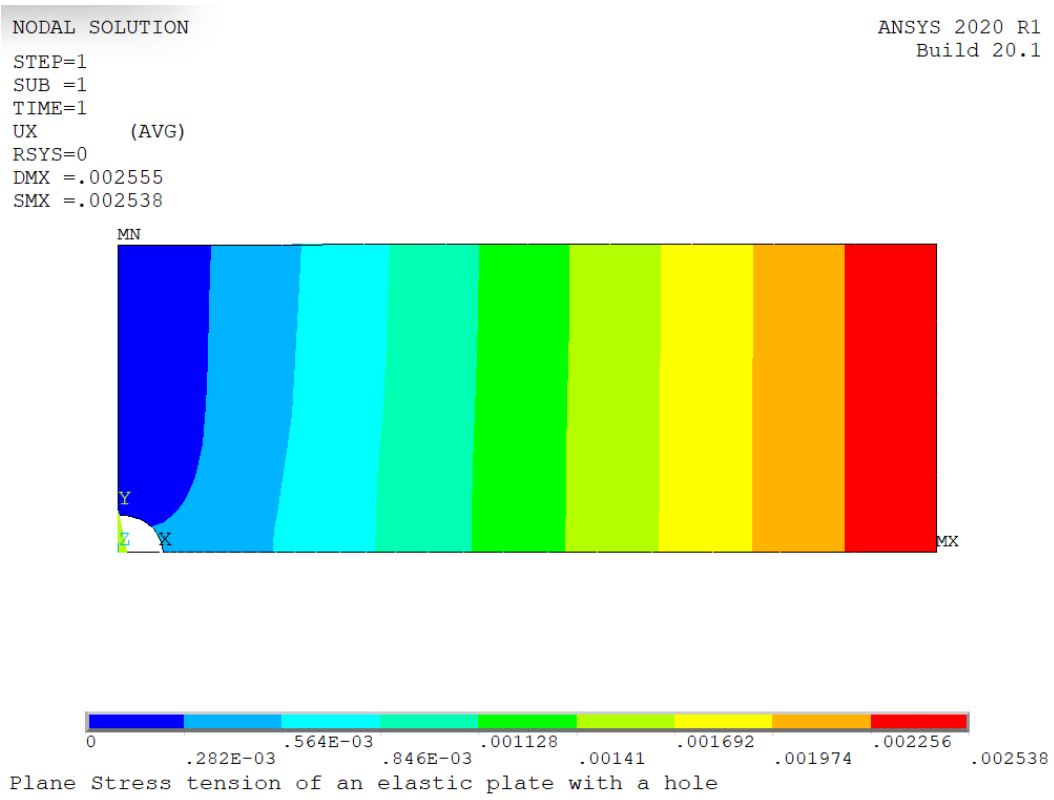


Fig. 9. Distribution of the displacements U_x

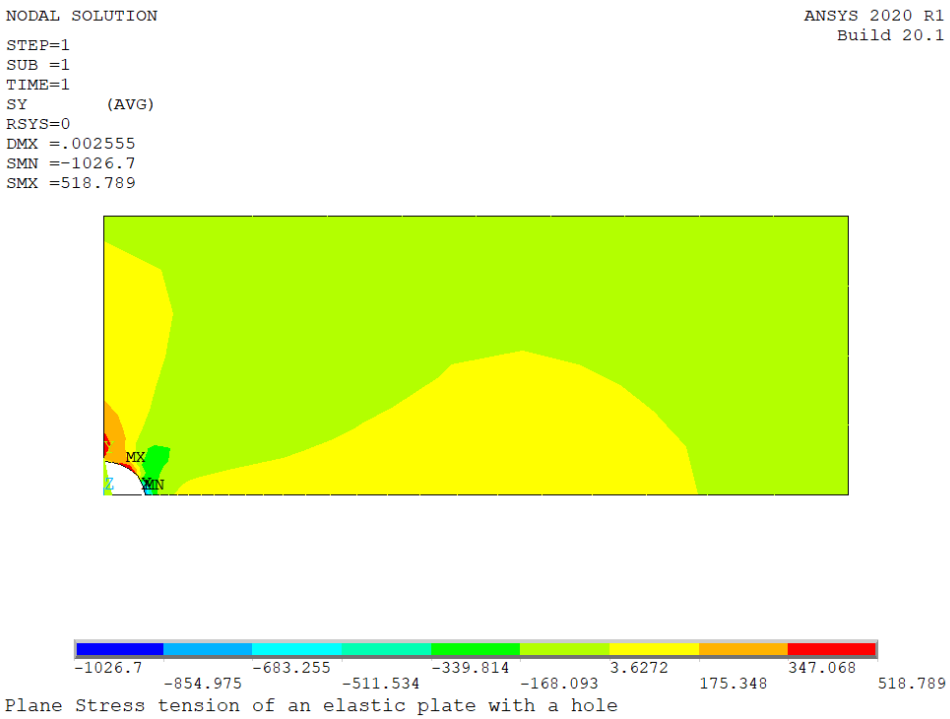


Fig. 10. Distribution of axial stresses T_{yy}

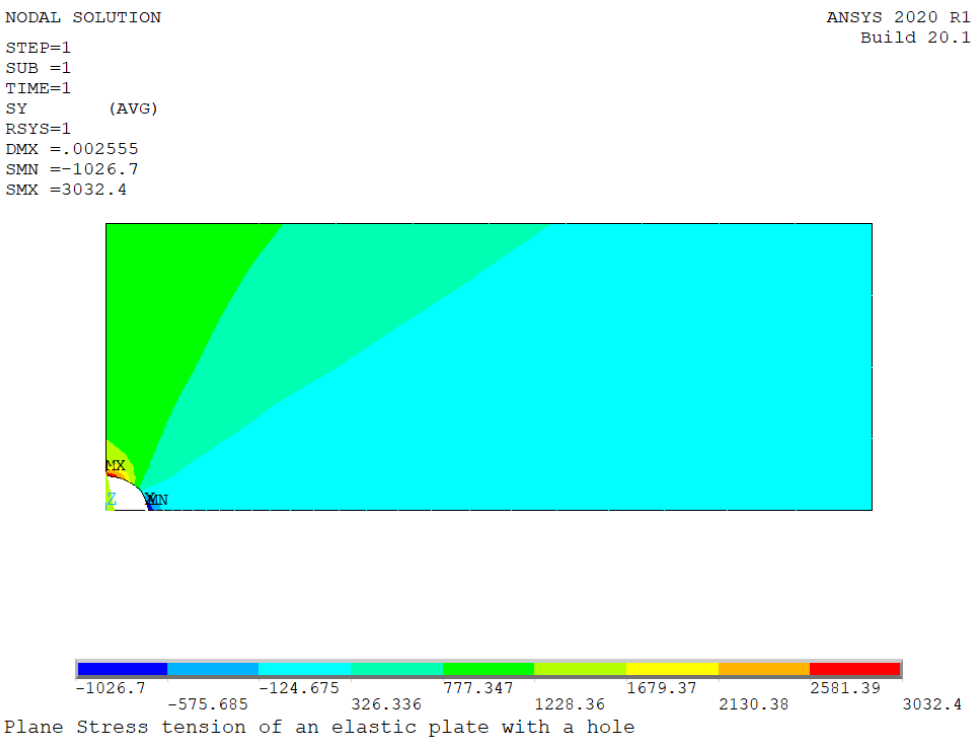


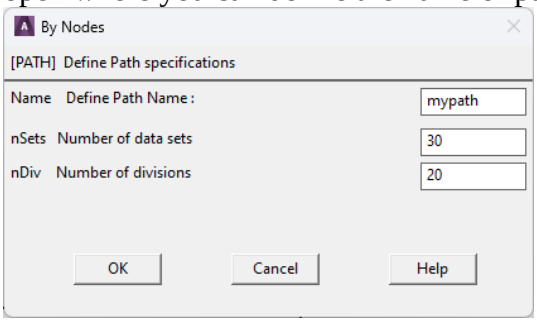
Fig. 11. Distribution of tangential stresses $T_{\theta\theta}$

As can be seen from Fig. 10 and 11, the hole is the stress concentrator.

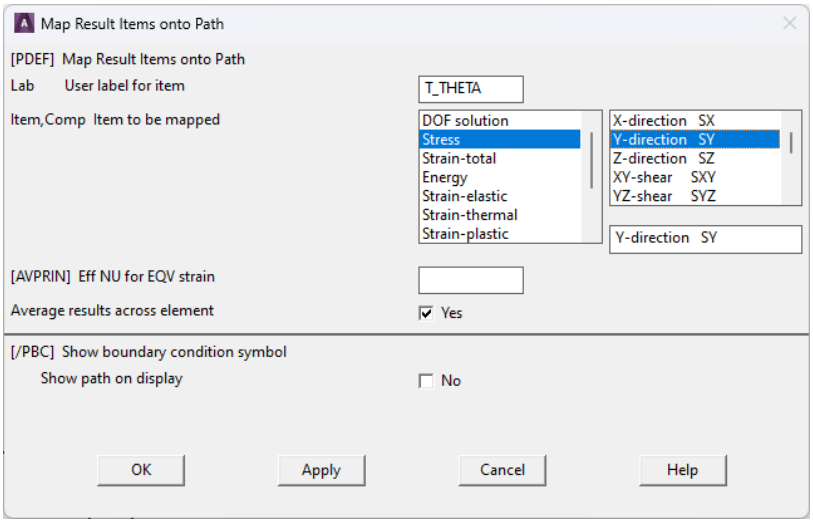
This test problem has analytical solution. Let us compare this analytical solution with our numerical solution. In order to do this, let us plot a graph of tangential stresses along the path (Fig. 12) through the point (0,R) to point (0,B).

In interactive mode, you should follow the steps below:

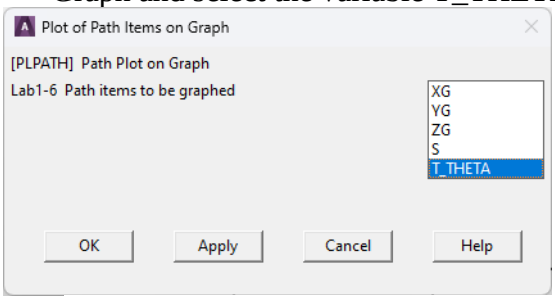
1) **Define the points for the path**. Plot Elements or Nodes (Utility Menu: Plot → Elements or Plot → Nodes), then in Main Menu select General Postproc → Path Operations → Define Path → By Nodes and pick two points with coordinates (0,R) and (0,b). The window will open where you can define the name of path, for example mypath:



2) The variable in x-axis is the distance by default. Next you should define the variable to be plot for y-axis: General Postproc → Path Operations → Map onto Path where you define the variable name and choose the variable. For tangential stresses $T_{\theta\theta}$ let us define the name T_THETA and select: Stress, Y-direction SY (provided that the current result system is global cylindrical system RSYS 1).



3) Plot the path on graph: General Postproc → Path Operations → Plot Path Item → On Graph and select the variable T_THETA:



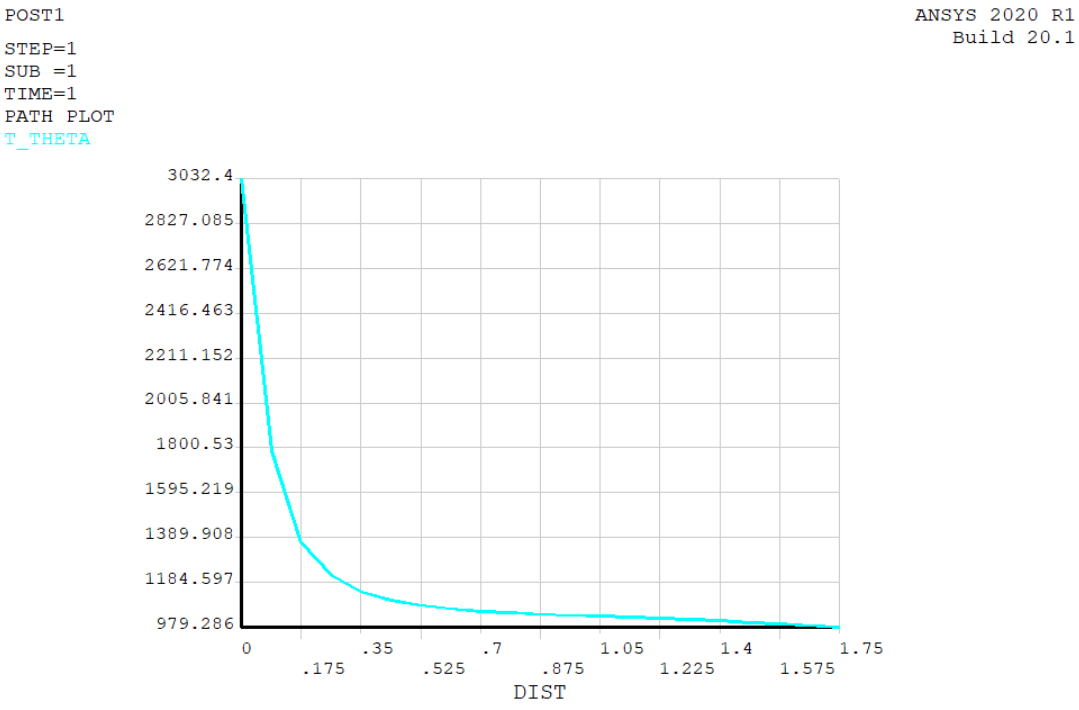


Fig. 12. Graph of tangential stresses along the path through the point (0,R) to point (0,B)

From Fig. 12 we can see that the maximum stress is in the point (0,R) and equals $\sigma_{\theta\theta} = 3032$ (kgf/cm^2). Thus, as the applied pressure load was p=1·10^3 (kgf/cm^2), our computations confirm the analytical result that the stresses around concentrator point increase approximately 3 times.

The following commands give Fig. 8-10:

```
/POST1
/PLOPTS,LOGO,OFF   ! Don’t show ANSYS logo
/PLOPTS,FRAME,OFF  ! Don’t show frame
/PLOPTS,DATE,OFF   ! Don’t show date

SET,last ! Read the last step
PLDISP,2 ! Plot deformed mesh with undeformed edge
! Delay for viewing the picture
*ASK,TMP,ANY NUMBER OR PRESS "ENTER"
PLNSOL,U,X !  Plot displacements ux
*GET,UXMAX,PLNSOL,0,MAX ! Get maximal value of displacements ux
! Delay for viewing the picture
*ASK,TMP,ANY NUMBER OR PRESS "ENTER"

RSYS,1     ! Set results coordinate system to global cylindrical
PLNSOL,S,Y ! Plot tangential stresses (in global cylindrical system)
```

Fig. 12 can be obtained using the following commands:

```
/POST1
! ******************************************************
! Graph of tangential stresses T_Theta
! along the path on OX-axis from point (0,R) to point (0,B)
! Delay for viewing the previous picture
*ASK,TMP,ANY NUMBER OR PRESS "ENTER"
PATH,XX,2
PPATH,1,,0,R
PPATH,2,,0,B
PDEF,T_Theta,S,Y
PLPATH,T_Theta
!******************************************************

RSYS,0     ! Set results coordinate system back to global cartesian
```

Note several new commands used the script. ***ASK** command enables us to make a pause to view the previous picture before plotting the next picture. ***GET** command sets a user-defined parameter **UXMAX** with the value of maximal displacement UX. Command **RSYS,1** sets the current coordinate results coordinate system to global cylindrical (default is cartesian system) and enables us to view results, such as tangential stresses $\sigma_{\theta\theta}$, in cylindrical system. The commands **PATH, PPATH, PDEF, PLPATH** give the graph of a variable along the defined geometric path.