TWO-DIMENSIONAL PROBLEM OF THE THEORY OF ELASTICITY. INVESTIGATION OF STRESS CONCENTRATION FACTORS.

1. INTRODUCTION

Two-dimensional problem of the theory of elasticity is a particular case of the 3-D problem. This problem can be solved as a plane elastic region with known boundary conditions (static or kinematic) and mass forces acting inside. The analytical solutions are known only for some simple cases as:

- thin plate of any shape in plane stress conditions (*Plane stress*)
- prismatic solid with assumed zero displacements in perpendicular direction to the section area (*Plane strain*)
- body of revolution loaded axis-symmetrically (*Axial symmetry*)

Each of these problems can be solved by using Finite Element Method. The discretization encloses plane representative region of the object. It should be remembered that the finite elements used in analysis have to correspond to Hooke's law formulation adequate to plane stress, plane strain or axis-symmetry type of problem.

2. PROBLEM DESCRIPTION

The work task is to analyze stress distribution in a thin plate made of aluminum alloy. The plate has oval opening and is subjected to uniform tensile stress applied on upper and lower edges (Fig.1). The numerical values of the stress concentration factors obtained from FEM-analysis should be compared with analytical values taken from the literature.

Data:

I – first personal parameter
N – second personal parameter

P *= (10000 +* ***I*****100) = ... [N]*
a *= (100+***N***) = ... [mm]*
r_1 *= (100+***N***) = ... [mm]*
r_2 *= (50+***I***) = ... [mm]*
M *= 800 [mm]*
h *= 300 [mm]*
δ *= 1 [mm] (thickness)*
E *= 7·10^4 [MPa]*
ν *= 0.32*

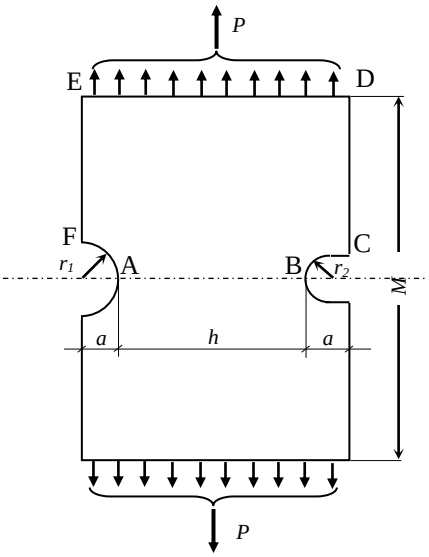


Fig.1. The notched plate

The task should compare the results obtained for different mesh densities (influence of discretization) and different types of finite elements (influence of approximation inside the element - shape function).

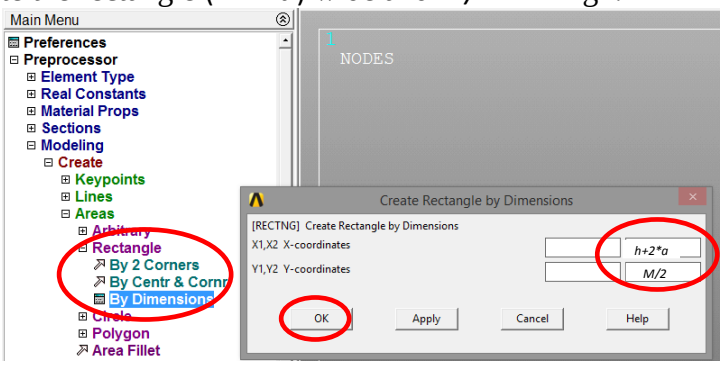
3. TYPICAL COURSE OF NUMERICAL ANALYSIS

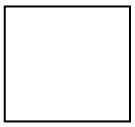
Taken into consideration that both load and shape of the plate are symmetrical, the model includes only a half part of the plate. Convenient units are: mm, N and MPa.

3.1. Preprocessor

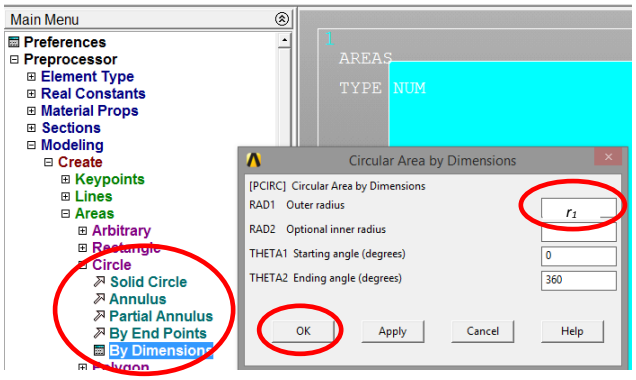
The construction of the solid model will be carried out using the Up_bottom technique, i.e. the so-called primitives:

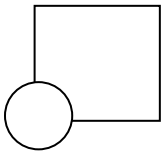
a) Create the rectangle (***h+2*a***) wide and ***M/2*** *mm* high:



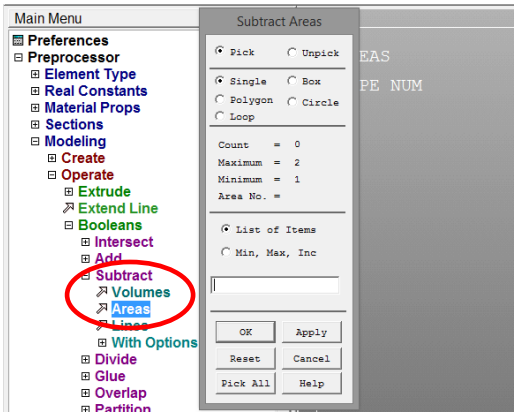


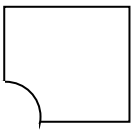
b) Create a circle r_1 in radius:



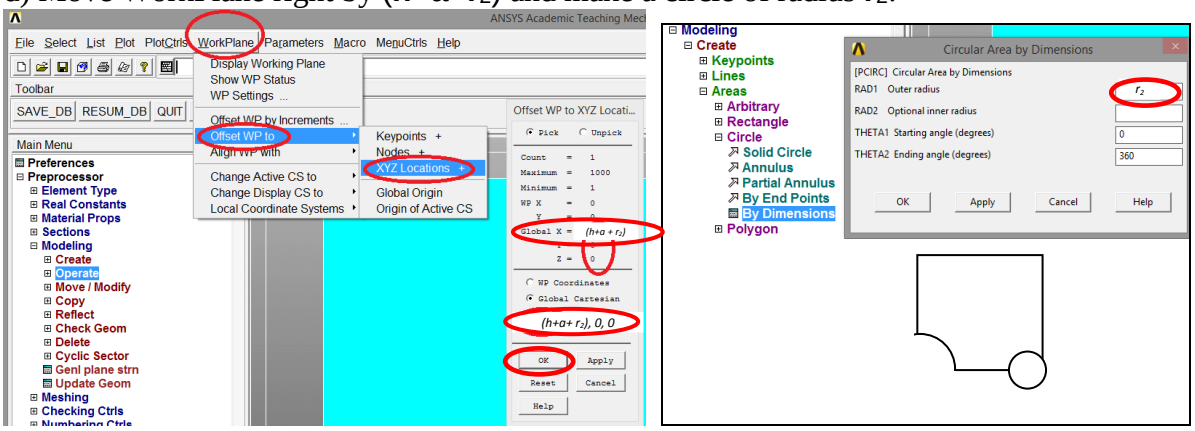


c) Subtract the circle from the rectangle:

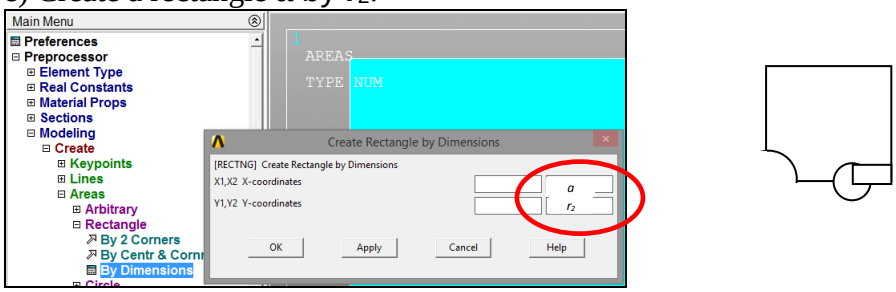




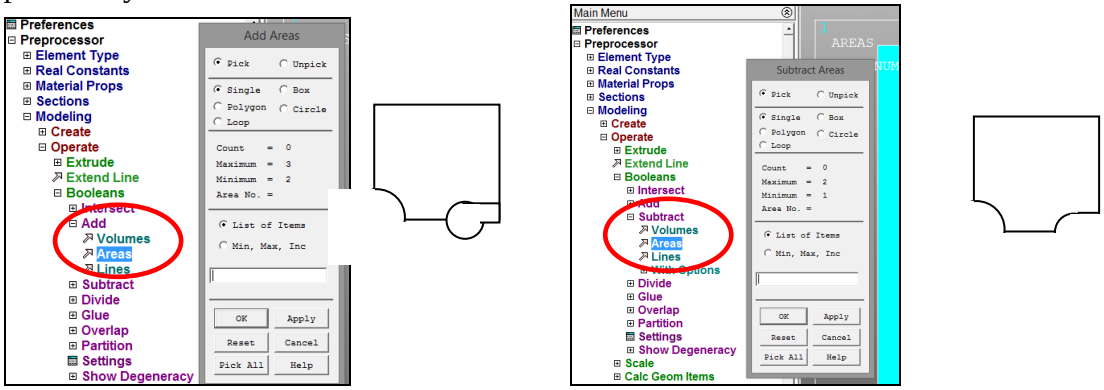
d) Move WorkPlane right by (**h**+**a**+**r**$_2$) and make a circle of radius **r**$_2$:



e) Create a rectangle **a** by **r**$_2$:

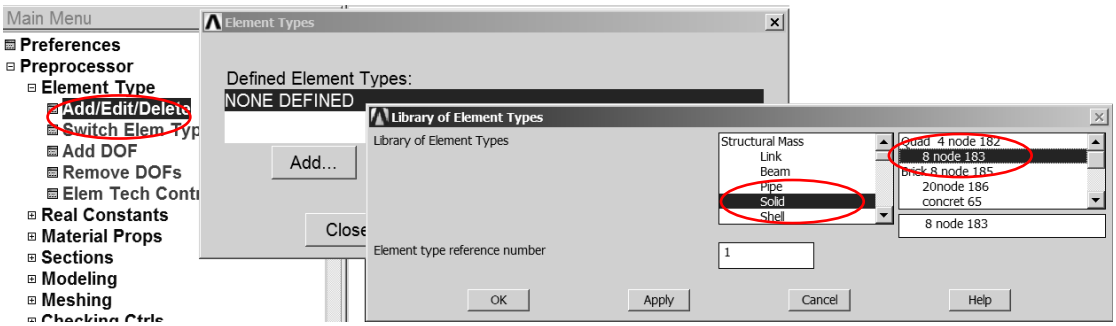


f) Add a circle and a rectangle together, and then subtract the resulting figure from the previously built area:

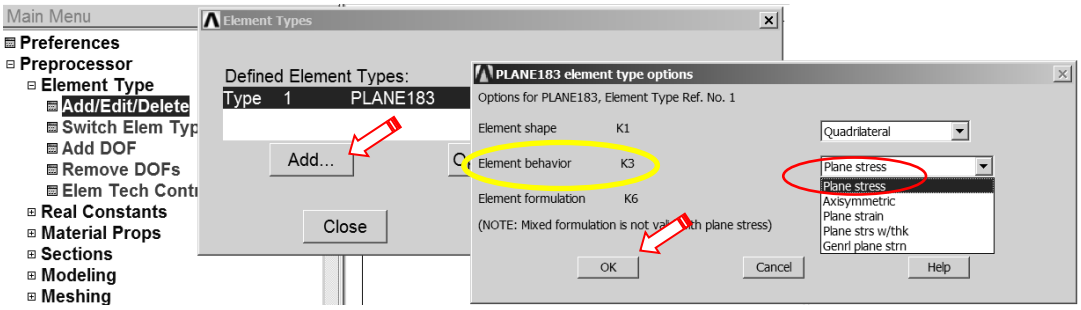


g) Draw the area lines:

Chose the element type (eight node PLANE183 or four node PLANE182) and plane stress behaviour (*Plane stress*):
ATTENTION: *for Solid 182 choose „Enchanced strain”*



Choosing element type



Setting element options

Define elastic isotropic model of the material by the two constants: Young modulus (*EX*) and Poisson ratio (*PRXY*):

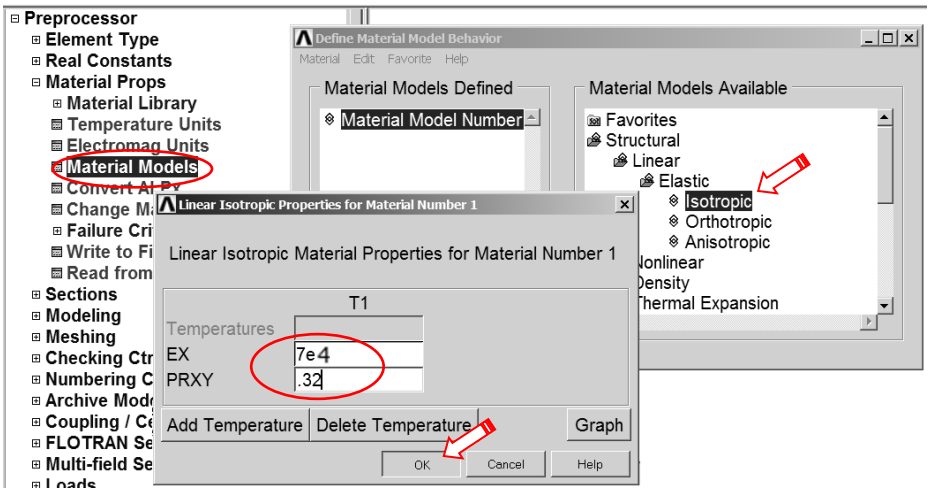


Fig. 15. Defining material properties

Determination of discretization density:

In the example, we define division densities on successive outer lines of the analyzed area.

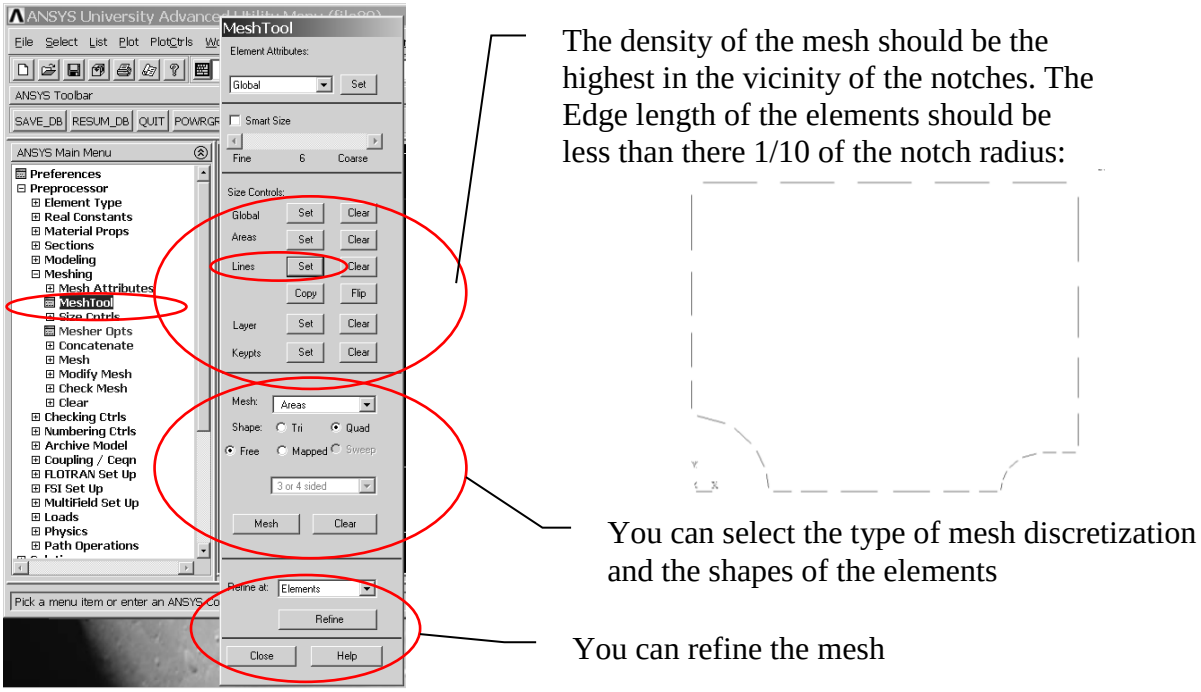
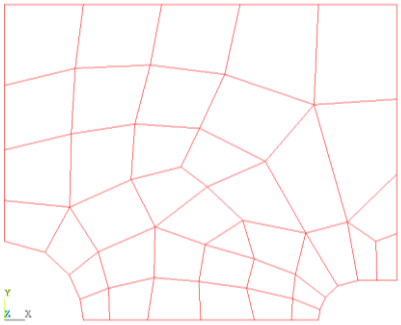


Fig. 2. Determining the parameters for controlling the mesh density

You can control the direction of density on the lines using ***SPACE*** parameter.



Podział na elementy skończone (np. *free meshing*).

Fig. 4. Finite element mesh in the shield area

3.2. Solution – solver module

Determination of the support conditions:

a) condition of symmetry on the line connecting the bottoms of the notches *(taking away the freedom of displacement in the y direction),*

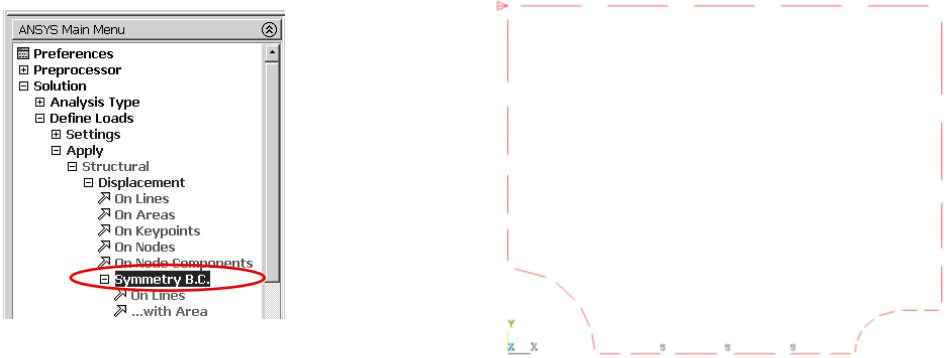


Fig. 5. Determination of symmetry conditions on lines

b) additionally, the possibility of displacement in the x direction of any selected node must be denied (you can enter this condition directly into a node or Keypoint).

Apply the surface load as negative pressure on line ED:

$$p = -P/(h+2*a)/\delta \; [MPa]$$

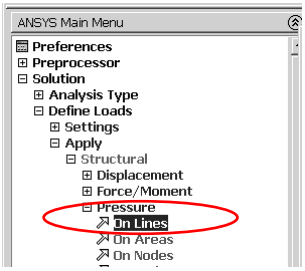
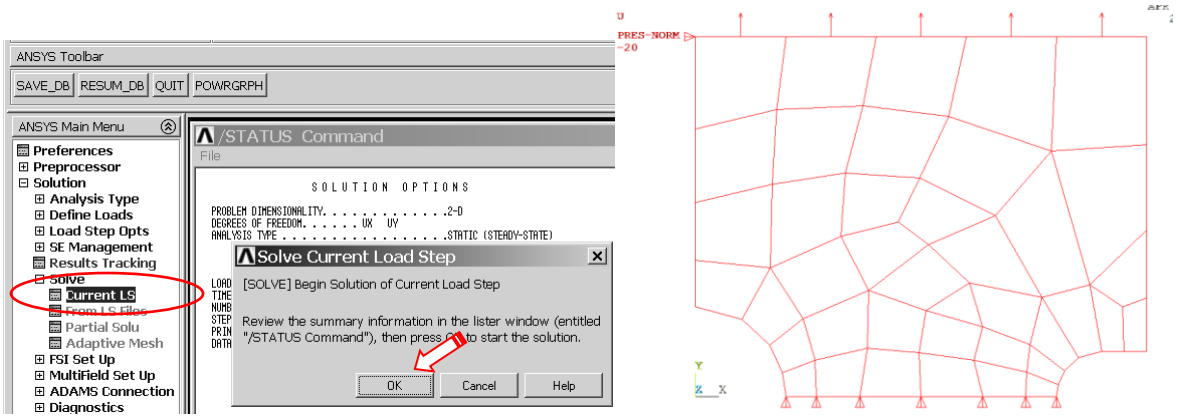


Fig. 6. Applying load at line

Starting calculations.

Before starting the calculation process, it is worth saving the database. This can be done with the SAVE command in the window ANSYS Toolbar (see p. 3.2.1).

Run calculations with the command: *Solve* > ***Current Ls.***



Rys. 7. Uruchomienie procesu rozwiązania

3.3. General postprocessor

Presentation of results in the form of contour maps.

a) Plot the stress distribution in Y direction (SY):

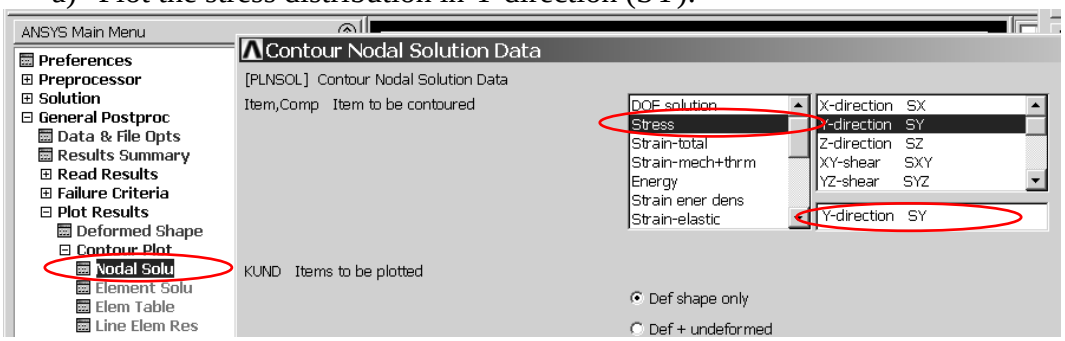


Fig. 8. Running the command to draw the SY component map of the state of stress

b) Show the plot of Von Mises stresses (von Mises SEQV).

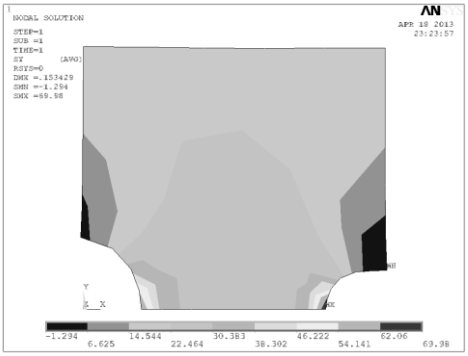


Fig. 9. Distribution of Von Mises stress

Saving displayed plot to graphics file:

Each plot displayed in GUI Window can be copied to the graphics file

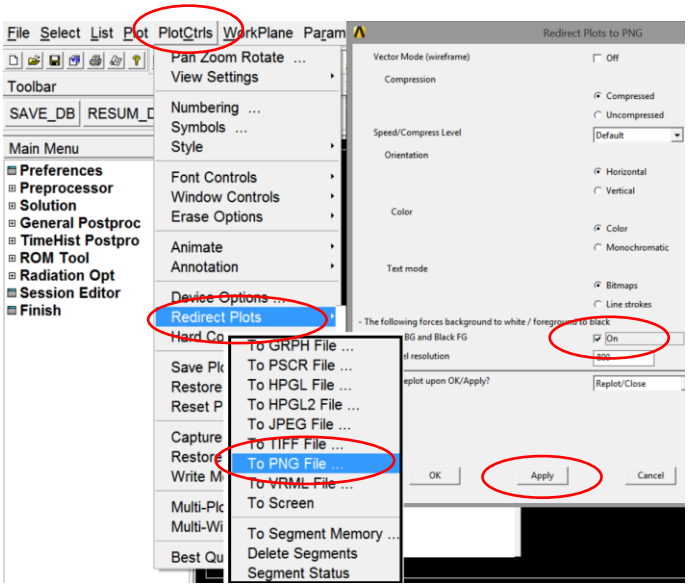


Fig. 10. Saving graphic files

Diagrams are used to present stress (components: SX, SY and SEQV) along the plane of symmetry:

a) Choose the path, where the argument s (distance) will be measured, by picking two nodes A i B):

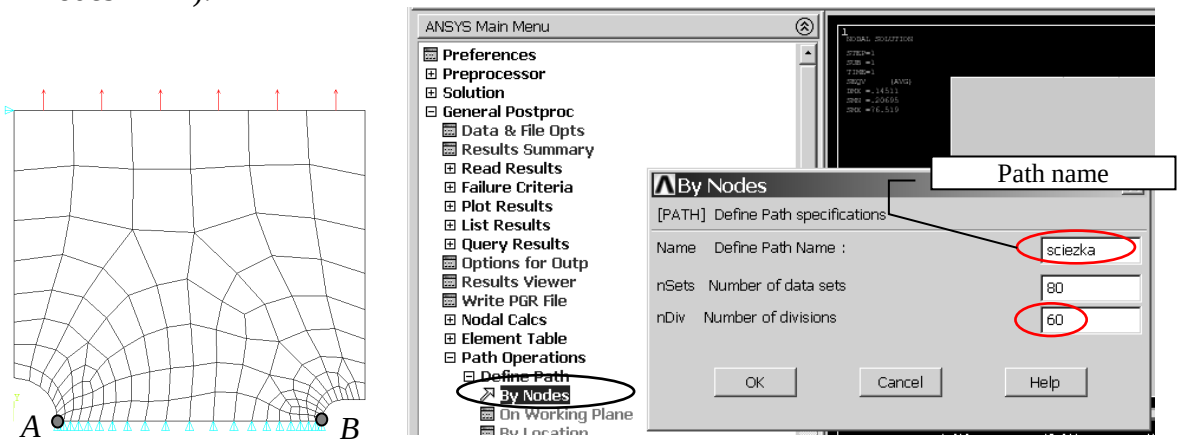


Fig 11. Defining path and path options

b) Choose needed function: SX(s), SY(s), SEQV(s). Each function can be named separately *(as User label for item)*, but this field does not need to be filled:

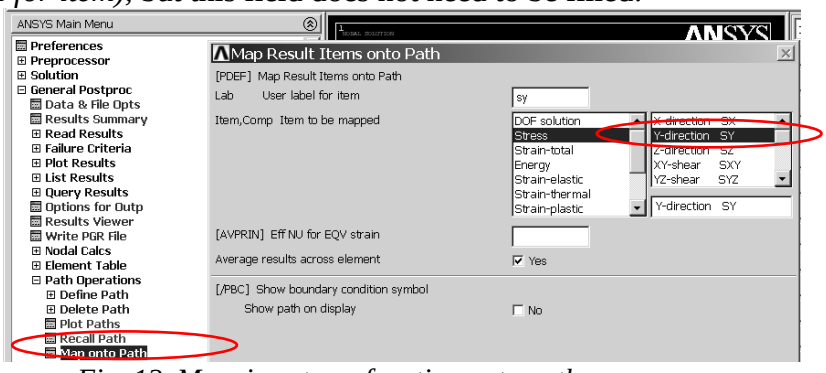


Fig. 12. Mapping stress function onto path

c) Plotting the diagram of chosen functions.
The scale of axes or lines colors can be changed in *Utility Menu (Plot Ctrls>Style>Graphs).*

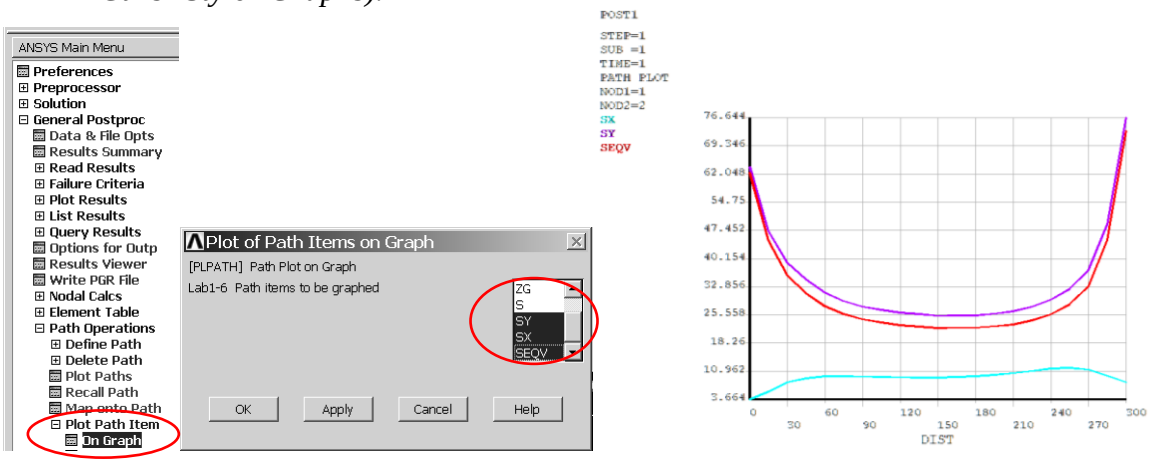


Fig. 13. Plotting path items on a graph

4. INTERPRETATION OF THE RESULTS. TASKS TO BE DONE

Compare results for:

a) Different mesh densities (discretisation influence):
- about 150 elements (**Mesh 1**),
- about 400 elements (**Mesh 2**),
- about 1500 elements (**Mesh 3**),

b) Different elemen types (aproximation influence)
- 8 noded elements *(Plane 183)*.
- 4 noded elements *(Plane 182)*,

Put the results into a **Table**:

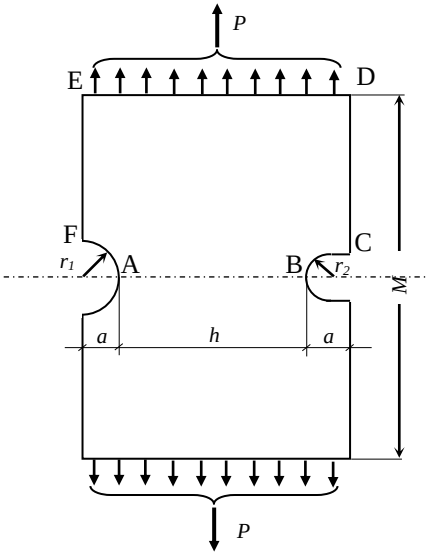
Number of nodes NN, number of elements NE, UY_{max}, SY_{max}^{A}, SY_{max}^{B}, SX^{A}, SX^{B}, $SEQV_{max}$, α_{FEM}^{A}, α_{FEM}^{B}, α_{T}^{A}, α_{T}^{B}, where:

SY_{max}^{A}, SY_{max}^{B} –	Maximum normal stress in Y at point A and B,
SX^{A}, SX^{B} –	Stress in X at point A and B,
$\alpha_{FE}^{A} = SY_{max}^{A} / \sigma_M$ –	stress concentration factor at the left notche (*punkt A*),
$\alpha_{FE}^{B} = SY_{max}^{B} / \sigma_M$ –	stress concentration factor at the right notche (*punkt B*),
$\sigma_M = P/(b-2a)/\delta$ –	Mean normal stress in the symmetry plane,
α_T^{A}, α_T^{B} –	Theoretical values of stress concentration factors taken from the literature.

Discuss the results.

Imię i Nazwisko: ……

I = (first personal parameter)
N = (second personal parameter)

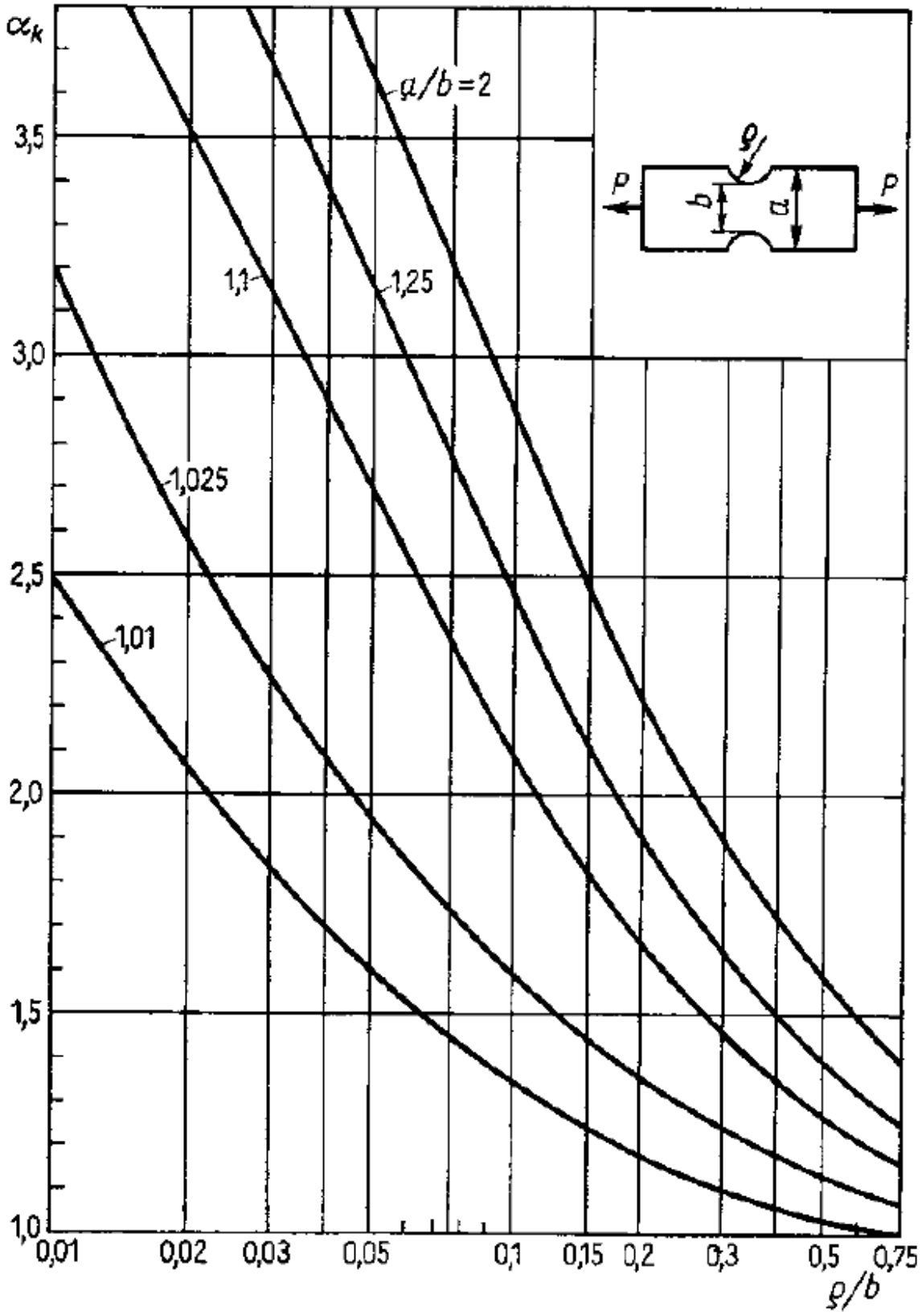


$P = (10000 + I*100) = \ldots\ldots\ldots\ [N]$
$a = (100+N) = \ldots\ldots\ldots\ [mm]$
$r_1 = (100+N) = \ldots\ldots\ldots\ [mm]$
$r_2 = (50+I) = \ldots\ldots\ldots\ [mm]$
$M = 800\ [mm]$
$h = 300\ [mm]$
$\delta = 1\ [mm]$ *(thickness)*
$E = 7 \cdot 10^4\ [MPa]$
$\nu = 0.32$

	8 noded elements (PLANE183)			4 noded elements (PLANE182)		
	Mesh 1	**Mesh 2**	**Mesh 3**	**Mesh 1**	**Mesh 2**	**Mesh 3**
No. of nodes						
No. of elements						
UY_{max}						
SY_{max}^{A}						
SY_{max}^{B}						
SX^{A}						
SX^{B}						
$SEQV_{max}$						
α_{FE}^{A}						
α_{FE}^{B}						

$\sigma_M = P/(b-2a)/\delta =$	**Plots needed** ➔ (should be archived during program session)	1) FE mesh. 2) UY(x,y) 3) SY(x,y) 4) SX(x,y) 5) SEQV(x,y) 6) Graph:SY(s) ,SX(s),SEQV(s)	**Final report:** 1) Introduction 2) Assumptions for the modeling 3) model description *(solid model, mesh, boundary cond. and loads)* 4) Results 5) Results in the Table 6) Discursion 7) Conclusion
$\alpha_T^{A} =$			
$\alpha_T^{B} =$			

Conclusion:


α_k
3,5
3,0
2,5
2,0
1,5
1,0
0,01
0,02
0,03
0,05
0,1
0,15
0,2
0,3
0,5
0,75
ϱ/b
a/b=2
1,1
1,25
1,025
1,01
P
P
b
a
ϱ