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:

b=*500mm*, ***h***=*800mm*, $\boldsymbol{\delta}$=*2mm (thickness)*,
$\boldsymbol{r_1}$=*25mm*, $\boldsymbol{r_2}$=*50mm*, ***a***=*60mm*,
$\boldsymbol{E}=7\cdot10^4$ *MPa*, $\boldsymbol{\nu}=0.32$
P=*20kN*

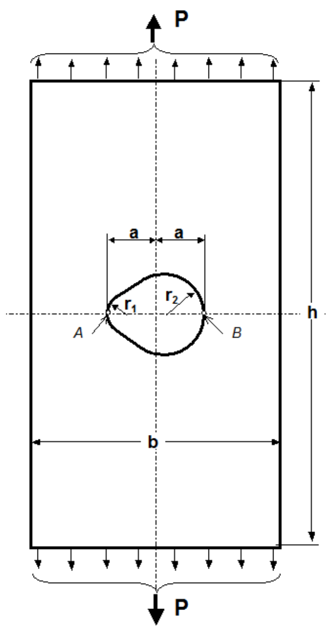


Fig.1. The notched plate

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 solid model is built “from the Top Down” by making use of primitives.

a) Create the rectangle 500 wide and 400 mm high:

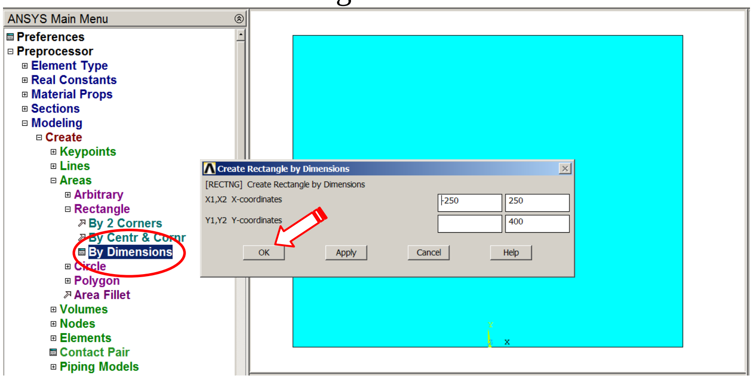
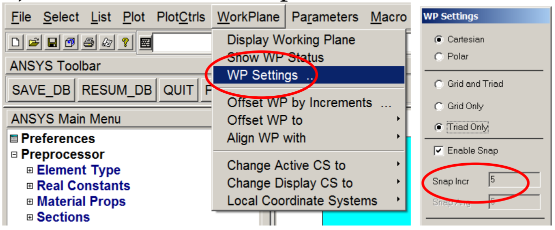


Fig. 2. Creation of the rectangle

b) Set WorkPlane snap increment to 5mm and offset Workplane two snaps right:



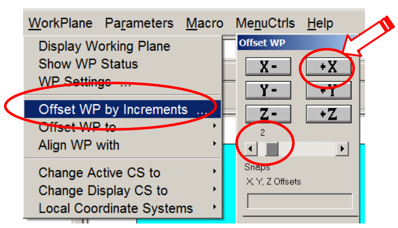


Fig. 3. Setting WorkPlane snap increment *Fig.4. Offset of the WorkPlane two snaps right*

c) Create semicircle: radius $\mathbf{r_2}$=50mm at WorkPlane origin:

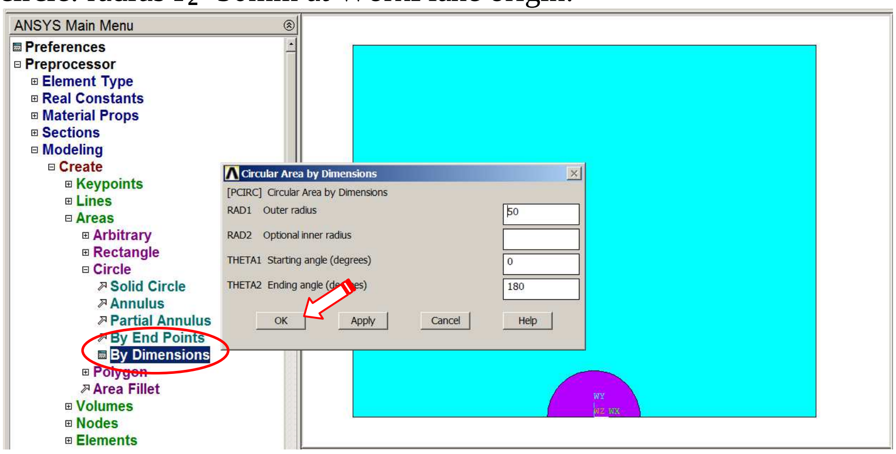


Fig. 5. Creation of $\mathbf{r_2}$ *semicircle*

d) Offset WorkPlane nine snaps left:

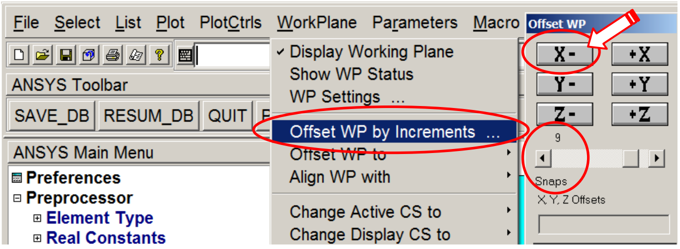


Fig.6. Offset of the WorkPlane nine snaps left

e) Create semicircle: radius $\mathbf{r_1}$=25mm at WorkPlane origin:

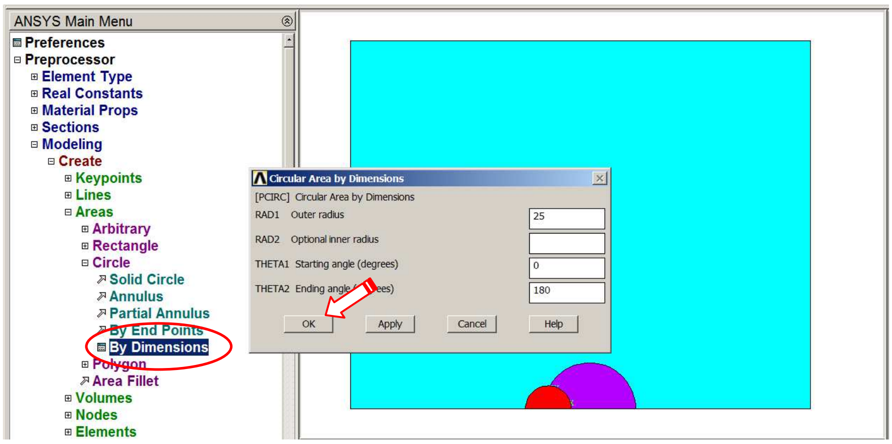


Fig.7. Creation of $\boldsymbol{r_1}$ semicircle

f) Plot lines:

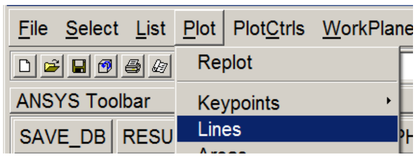


Fig.8. Plotting lines

g) Create line at angle to two lines:

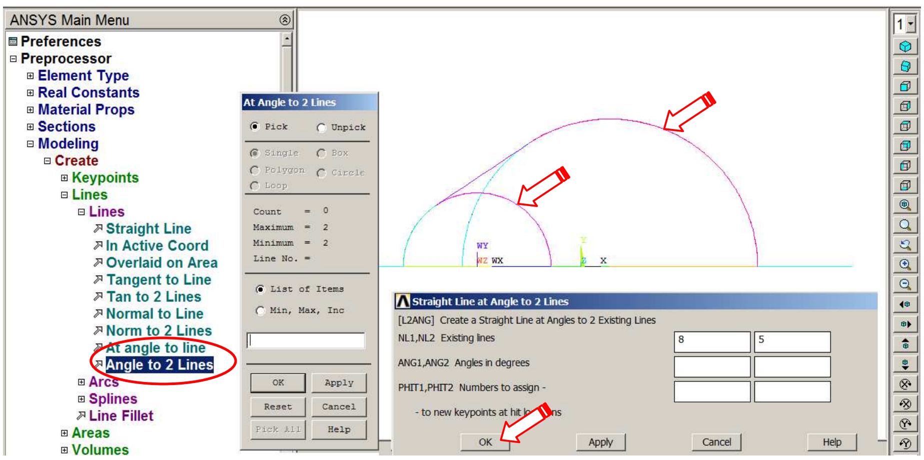


Fig.9. Creation of the tangent line

h) Create the area of the opening through keypoints:

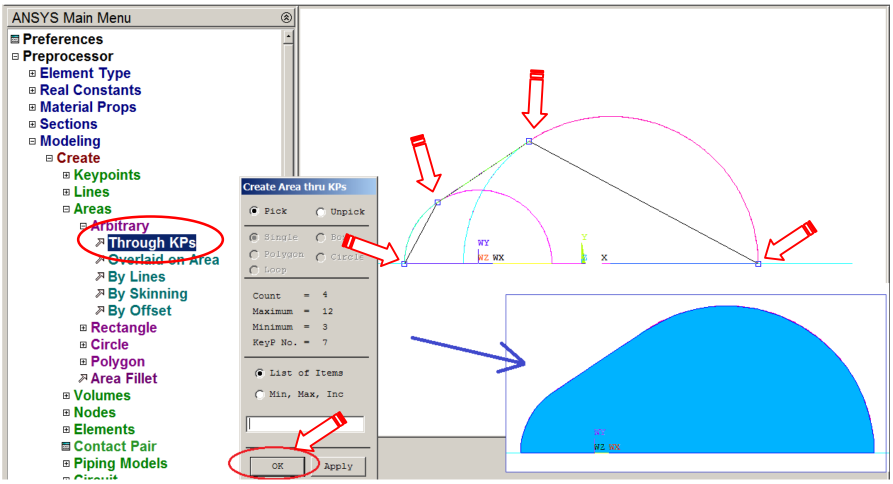


Fig.10. Creation of the area by keypoints

i) Delete semicircles *(together with lines and keypoints)*:

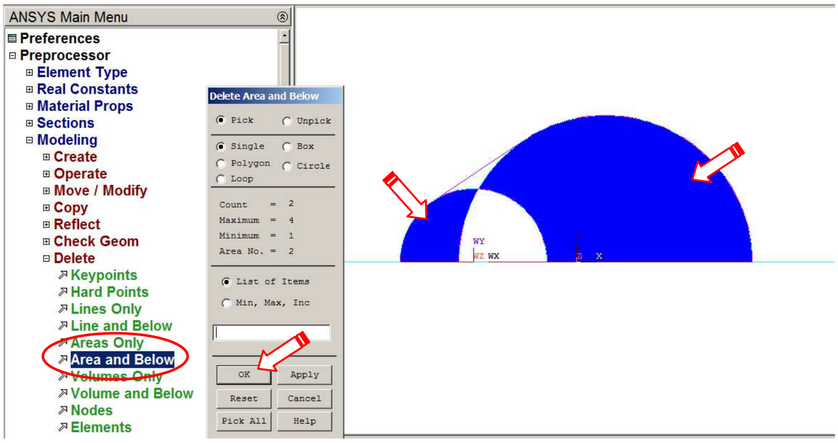


Fig11. Delete unnecessary areas

j) Subtract the area obtained in point h) from the rectangle:

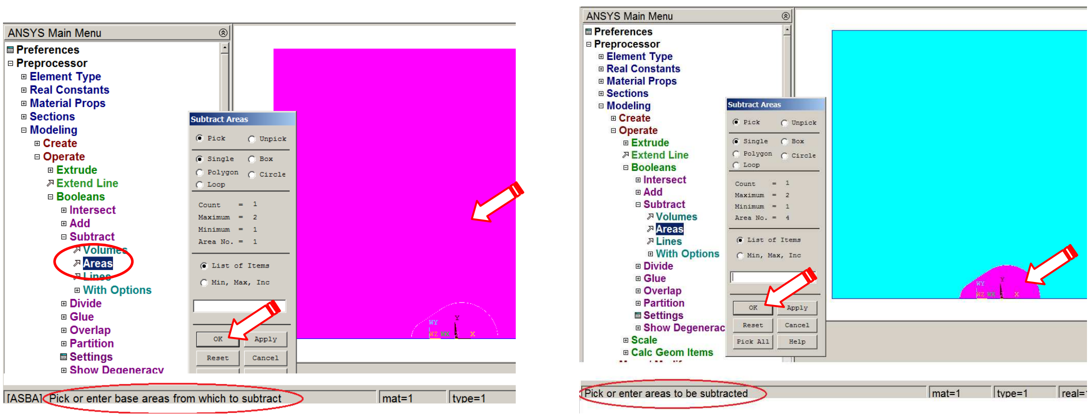


Fig.12. Substruction of the area created in point h) from the rectangle of point a)

<u>Chose the element type</u> (four node PLANE42 or eight node PLANE82) and plane stress behaviour (*Plane stress*):

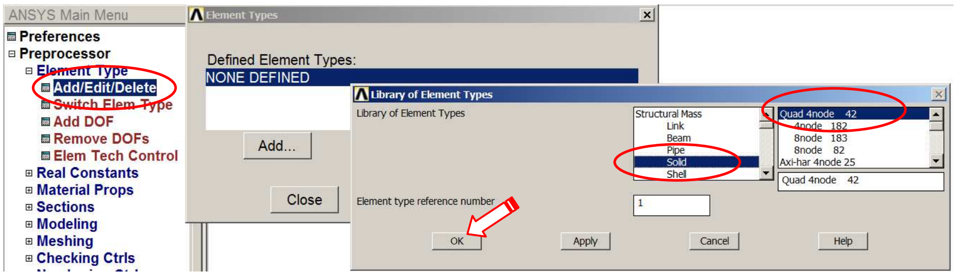


Fig.13. Choosing element type

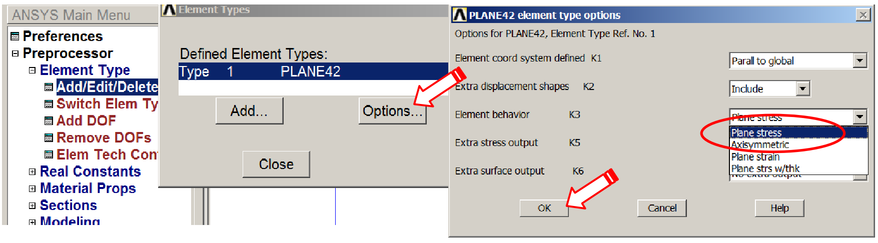


Fig.14. Setting element options

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

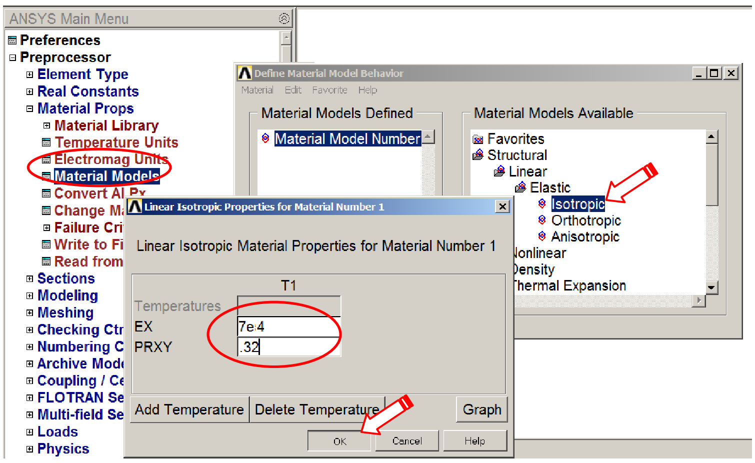


Fig. 15. Defining material properties

<u>Define mesh density</u>:

The density of the discretization is defined at the edges of the plate. Size of elements in the region of predicted high stress (the bottom of the notch – point A and B in Fig.1) should be smaller than in other parts of the plate. The mesh density should be largest at the bottom of the notch and can be modified along the line by SPACE variable. The edges of elements in these regions should not be greater than 1/10 of the radius of the notch. There is also possibility to choose the shape of the element (triangle, quadrilateral) and the type of mesh (mapped or free).

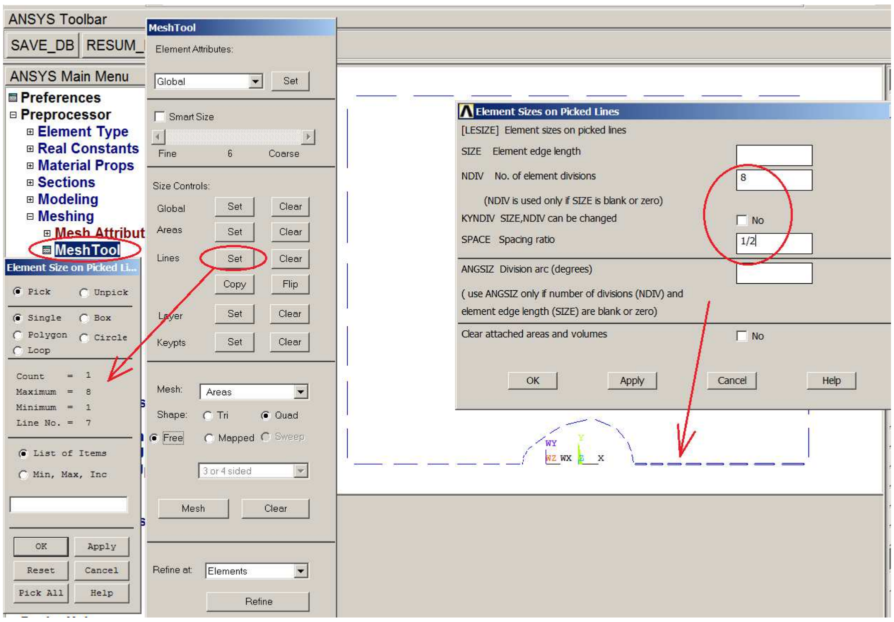


Fig. 16. Setting of size control on lines

<u>Mesh the area of the plate</u> (*free meshing*).

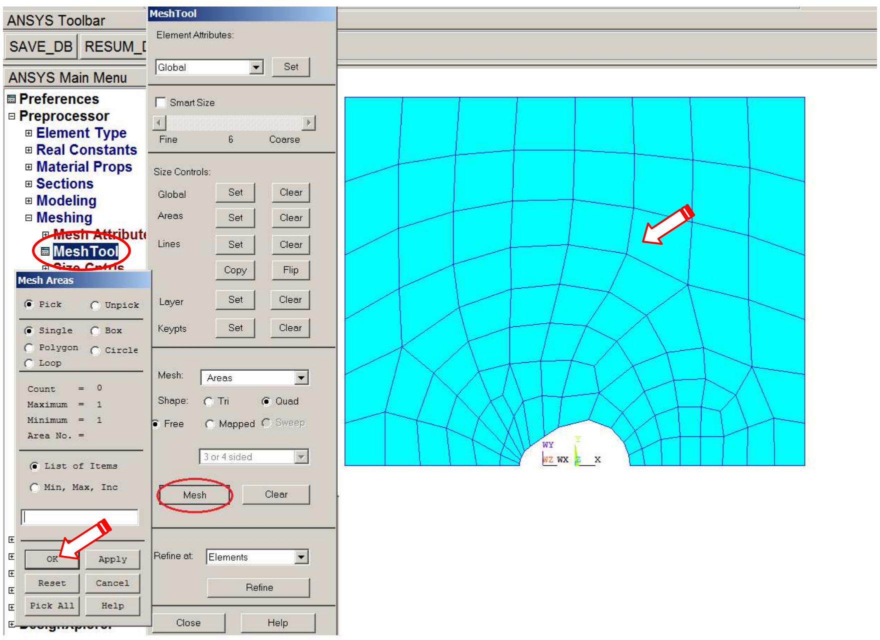


Fig. 17. FE mesh

3.2. Solution

<u>Defining boundary conditions</u>:

a) Symmetry on the lines in the symmetry plane (displacement constraints in y direction),

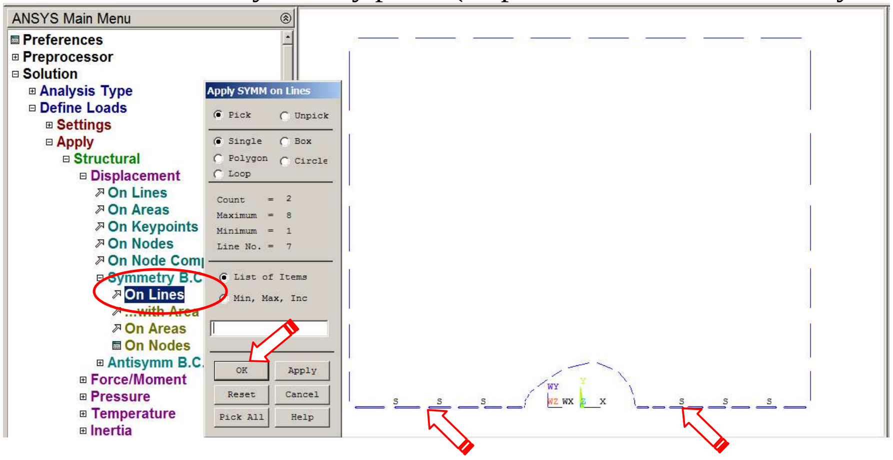


Fig. 18. Setting symmetry BC on lines

b) Constrained displacement in x direction on the arbitrary chosen keypoint,

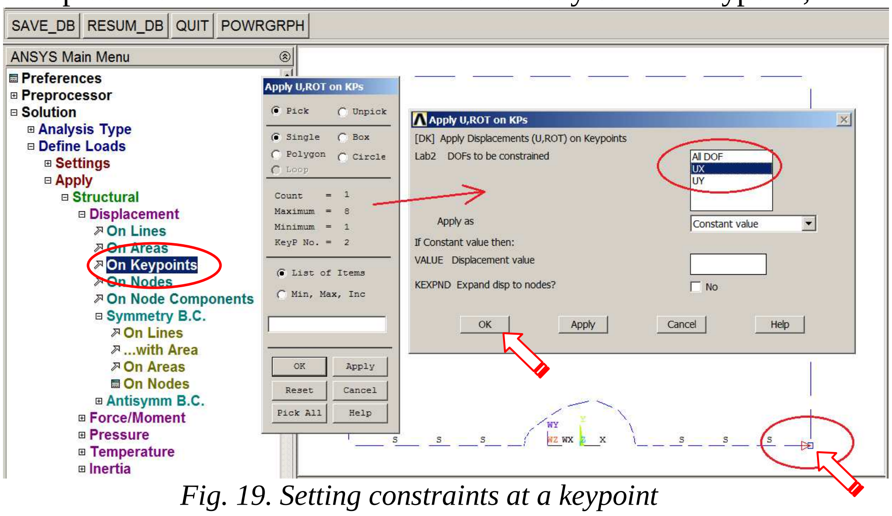


Fig. 19. Setting constraints at a keypoint

<u>Defining negative pressure on the top line</u>: *p= -20000/500/2 MPa*:

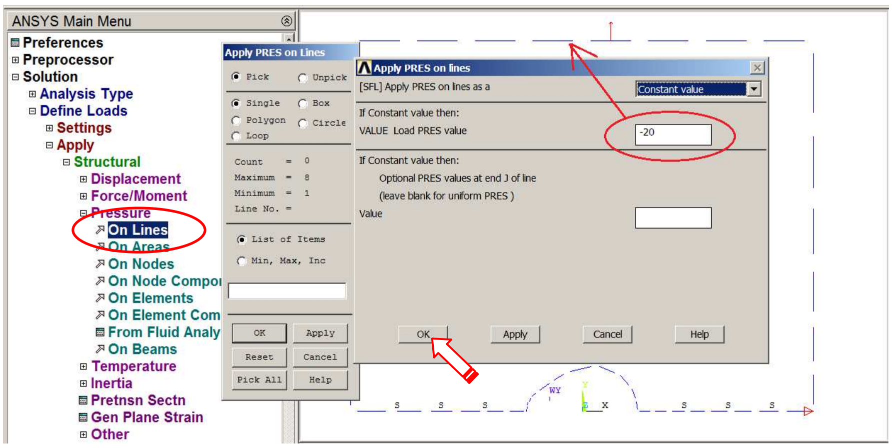


Fig. 20. Setting pressure on a line

Running solution.
Before running the process of solution it is recommended to save the datebase by SAVE command from menu File. The solving procedure is run by command: Solve/Current LS

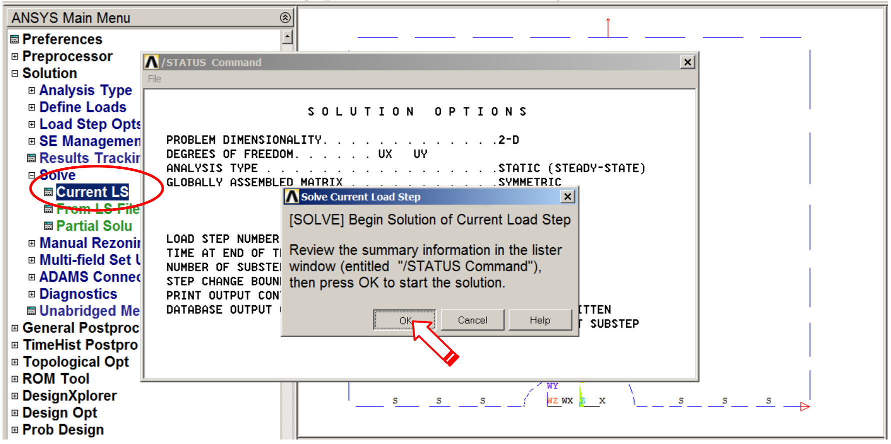


Fig. 21. Starting solution process

3.3. General Postprocessor

Present results in the form of maps:

a) Plot directional displacements UY (in Y):

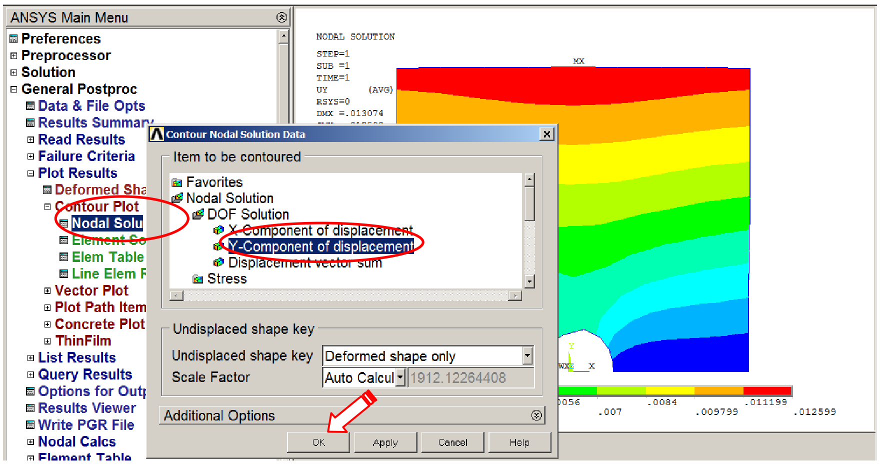
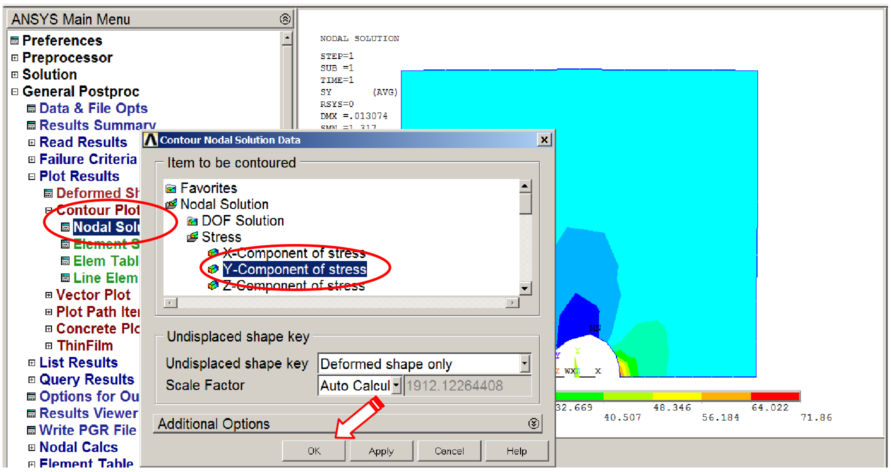


Fig. 22. Plotting UY displacement

b) Plot tensile stress component SY (in Y):



Rys. 23. Plotting SY stress

c) Plot Von Mises equivalent stress SEQV.

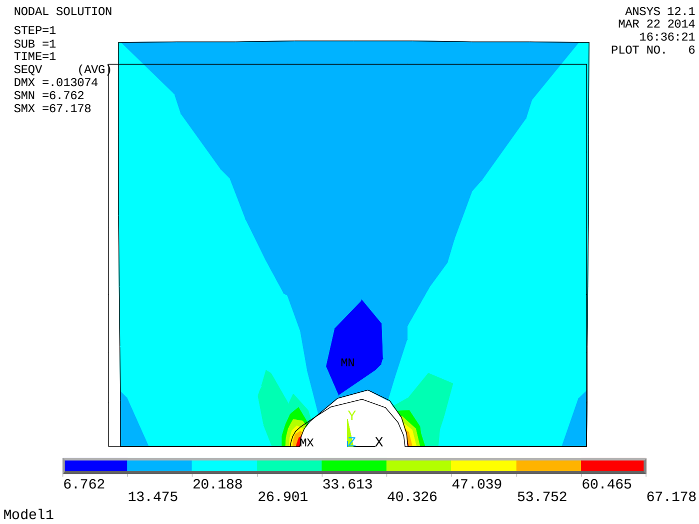


Fig.24. Von Mises stress distribution in the plate

<u>Saving displayed plot to graphics file:</u>

Each plot displayed in GUI Window can be copied to the graphics file (Fig. 25). After finishing the work with ANSYS the archived files are available directly (JPEG, TIFF formats) or can be edited by Display program (GRPH format).

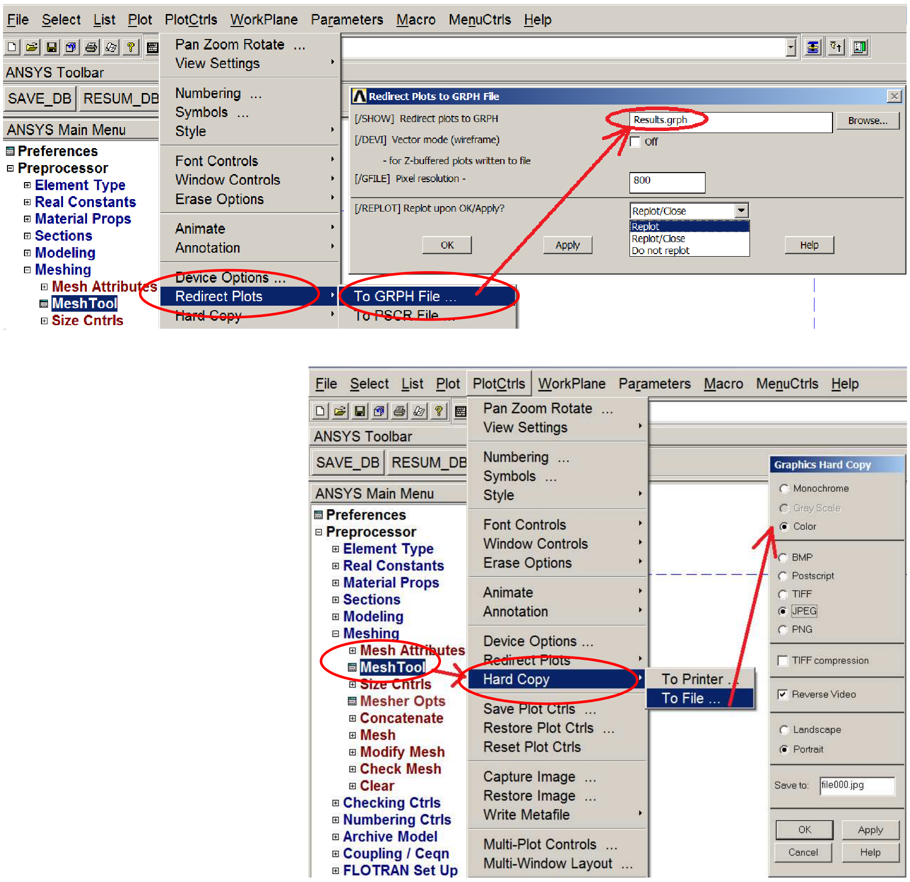


Fig. 25. Saving graphic files

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 four nodes (Fig 26):

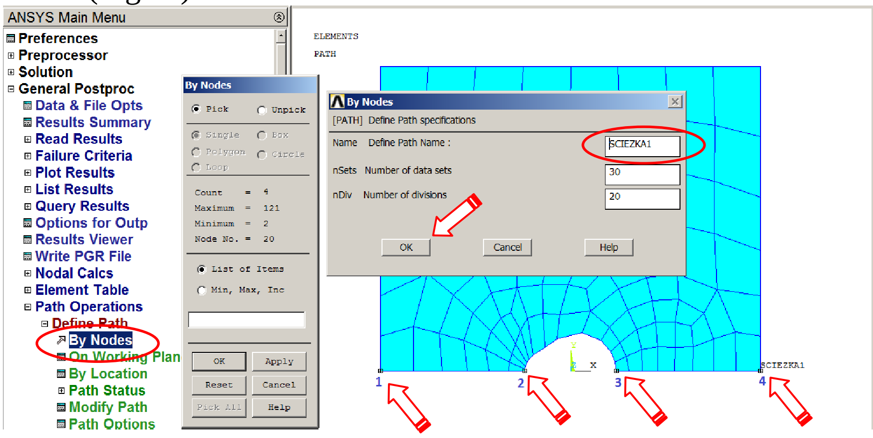


Fig. 26. Defining path and path options

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.

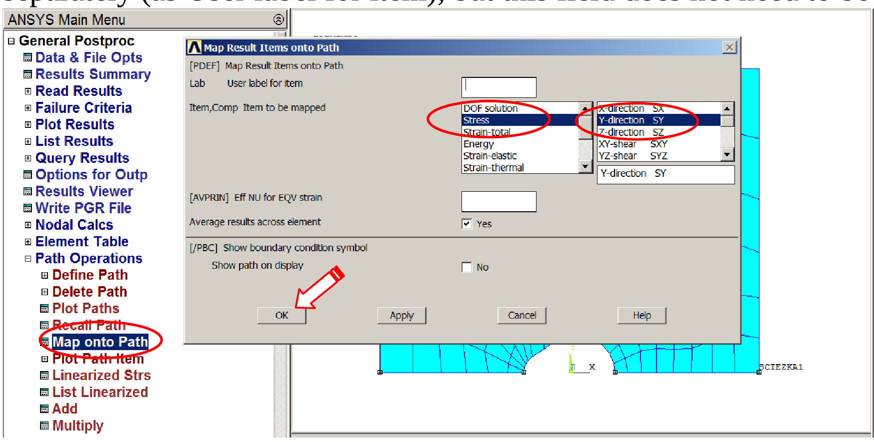


Fig. 27. Mapping stress function onto path

c) Plotting the diagram of chosen functions.

The scale of axes or lines colors can be changed in Utility Menu (Plot Ctrls>Style>Graphs).

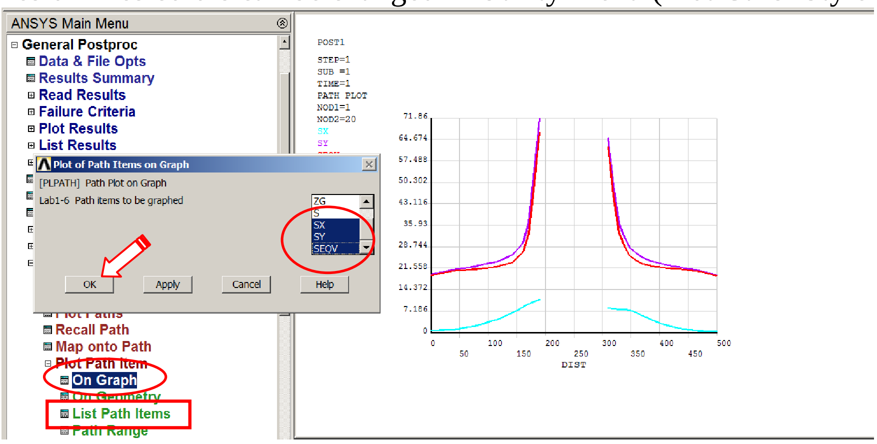


Fig. 28. Plotting path items on a graph

d) List the diagram of chosen functions *(command in box in Fig.28).*

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)
- 4 noded elements *(Plane 42)*,
- 8 noded elements *(Plane 82)*.

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.

	4 noded elements (**PLANE42**)			8 noded elements (**PLANE82**)		
	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}						

<table>
<tr><td>$\sigma_M = P/(b-2a)/\delta=$</td><td rowspan="3">Plots needed ➔
(should be archived
during program session)</td><td rowspan="3">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)</td><td rowspan="3">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</td></tr>
<tr><td>$\alpha_{T}^{A}=$</td></tr>
<tr><td>$\alpha_{T}^{B}=$</td></tr>
</table>

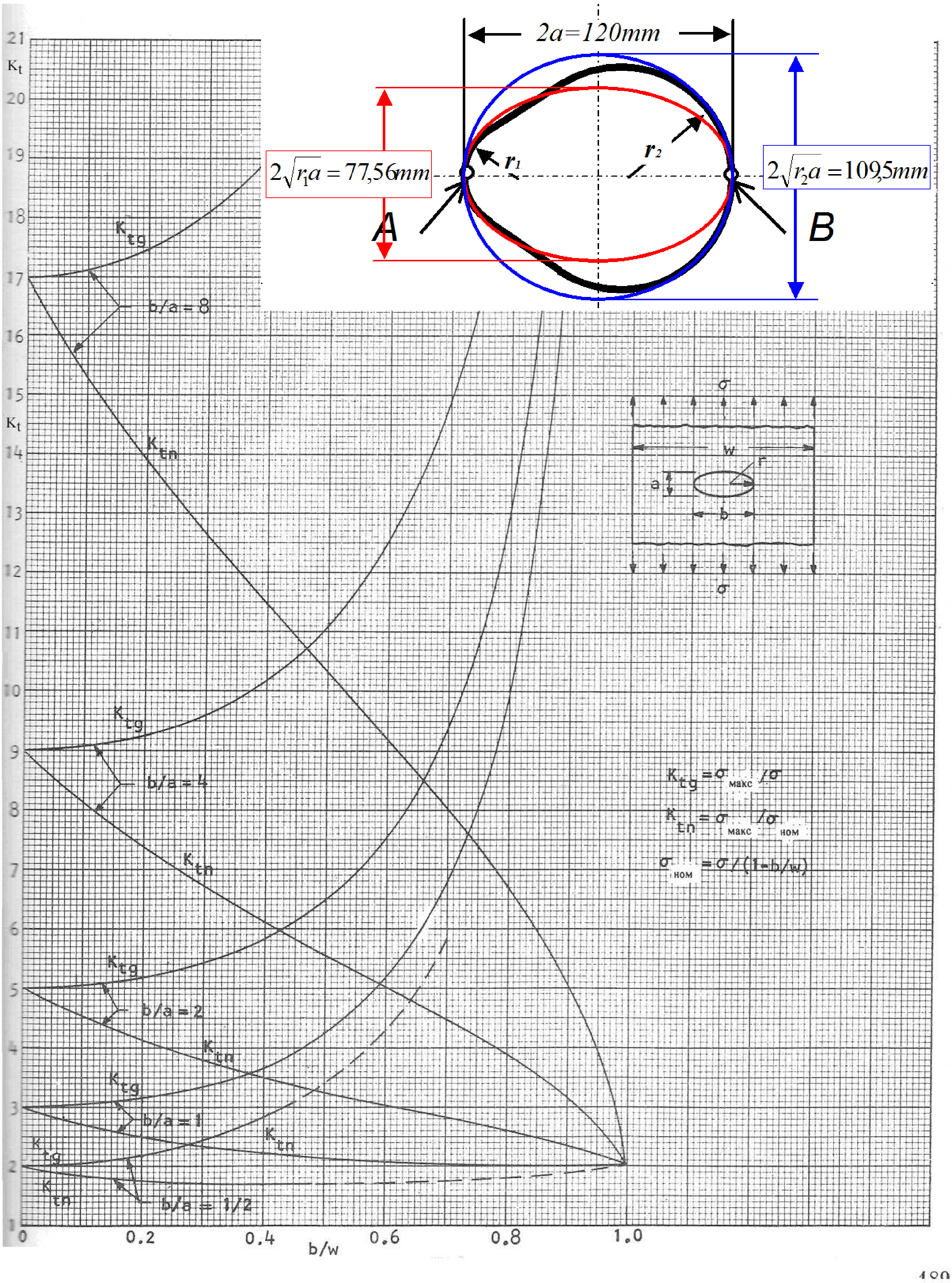


Fig. 29. Method of determining the stress concentration factor from the chart (α_T = K_{tn})