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, **\delta**=2mm (thickness), $r_1=25mm$, $r_2=50mm$, a=60mm, $E=7\cdot10^4$ MPa, v=0.32P=20kN

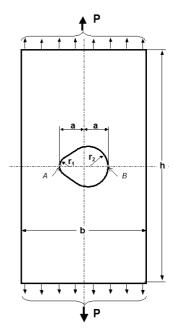


Fig.1. The notched plate

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:

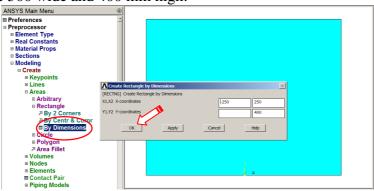


Fig. 2. Creation of the rectangle

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 r_2 =50mm at WorkPlane origin:

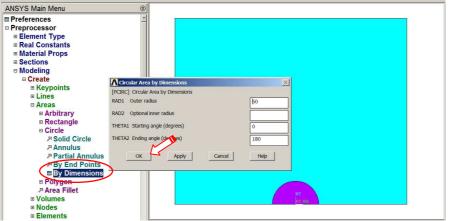


Fig. 5. Creation of r_2 semicircle

d) Offset WorkPlane nine snaps left:

<u>File Select List Plot PlotCtrls</u>	WorkPlane Parameters	Macro Offset WP
D 🚅 🖬 🖉 🖨 🖉 🌹 📰	✓ Display Working Plane	X- +X
ANSYS Toolbar	Show WP Status WP Settings	Y- +Y
SAVE_DB RESUM_DB QUIT		<u>Z-</u> +Z
ANSYS Main Menu	Offset WP by Increment	, , , , , , , , , , , , , , , , , , ,
Preferences	Align WP with	• Snaps
Preprocessor Element Type	Change Active CS to	V, Z Offsets
Real Constants	Change Display CS to	•

Fig.6. Offset of the WorkPlane nine snaps left

e) Create semicircle: radius r_1 =25mm at WorkPlane origin:

ANSYS Main Menu	8
Preferences Preprocessor Element Type Real Constants Material Props Sections Modeling	
 □ Create □ Keypoints □ Lines 	Cruder Area by Dimensions [PCIRC] Circular Area by Dimensions
	RAD1 Outer radius 25 RAD2 Optional inner radius
⊟ Circle २ Solid Circle २ Annulus	THETAL Starting angle (degrees) 0 THETA2 Ending angle (degrees) 180
Partial Annulus By End Points By Dimensions	OK Apply Cancel Hep
[®] Po lygon ,त्र Area Fillet ® Volumes ® Nodes ® Elements	

Fig.7. Creation of r_1 semicircle

f) Plot lines:

<u>File Select List</u>	<u>P</u> lot	Plot <u>C</u> trls	<u>W</u> orkPlane
□ ≥ ∎ @ @ @	Re	plot	
ANSYS Toolbar	Ke	points	•
SAVE_DB RESU	Lin	es	۶F
	۸		

Fig.8. Plotting lines

g) Create line at angle to two lines:

ANSYS Main Menu Preferences Preprocessor Belement Type	**************************************			1
Real Constants			•	6
Material Props	At Angle to 2 Lines			E
Sections	@ Pick C Unpick			6
Modeling	G Single C Box			
Create	CONTRACTOR CONTRA			
Keypoints	C Pelygon C Circle			
Lines	(noop		1	
Lines	Count = 0			
Straight Line	Maximum = 2			
	Minimum = 2	WY Y		
P Overlaid on Area	Line No. =	WX X		
Tangent to Line	C List of Items			
		Straight Line at Angle to 2 Lines		
	C Min, Max, Inc	[L2ANG] Create a Straight Line at Angles to 2 Existing Lin	nes	-
Norm to 2 Lines		NL1,NL2 Existing lines	8 5	
At angle to line	P			
Angle to 2 Lines		ANG1,ANG2 Angles in degrees		
Arcs	OK Apply	PHIT1,PHIT2 Numbers to assign -		
Splines	Reset Cancel	- to new keypoints at hit		
	Pick All Help	· to new keypoints at nice with		
Areas	FICK XII Help	- OK Apply	Cancel Help	-17
Volumes				-

Fig.9. Creation of the tangent line

Ex_1_2D Plate.doc

h) Create the area of the opening through keypoints:

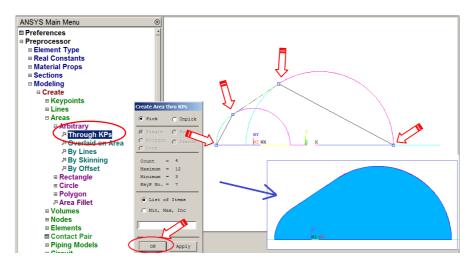


Fig.10. Creation of the area by keypoints

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

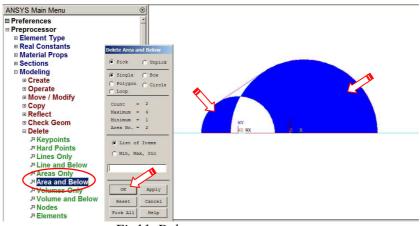


Fig11. Delete unnecessary areas

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

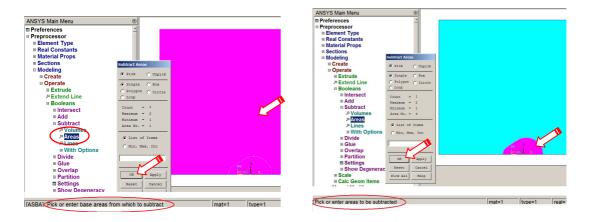


Fig.12. Substruction of the area created in point h) from the rectangle of point a)

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

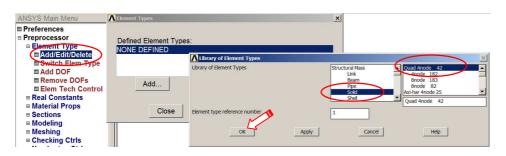


Fig.13. Choosing element type

ANSYS Main Menu	Clement Types		PLANE 12 element type opti	ons	X
Preferences			Options for PLANE42, Element Typ	e Ref. No. 1	
Preprocessor Element Type	Defined Element Types: Type 1 PLANE42		Element coord system defined K1		Paral to global 🔹
■ Add/Edit/Delete		M	Extra displacement shapes K2		Include
Switch Elem Ty	Add	Options	Element behavior K3	-	Plane suless
Remove DOFs Elem Tech Con	Close		Extra stress output K5		Axisymmetric Plane strain
Real Constants Material Props	Close		Extra surface output K6	N	Plane strs w/thk
Sections Modeling			ок	Cancel	Help

Fig.14. Setting element options

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

	8				
	-				
_					
	🔨 Define Material Model B				_ 🗆 🗙
s	Material Edit Havorite He	de			
	- Material Models	Defined	- Materia	al Models Available	
rary					
e Units	Material Model	Number 🖆			-
Units					
Py					
A Linear Tsotropic Pro	perties for Material Numb	er 1	×		
Linear Isotropic N	Aaterial Properties for	or Material Nun	nber 1 📊		
	T1				-1
Таниналар					
EX (
PRXY	32				
A	Delete Temperet	🔨	Onesh		
Add Temperature	Delete Temperatu		Graph		
		General	Helo		
	UK		1.0.45		
	s aray e Units Units Units Units Units CEP Py Linear Isotropic Pro Linear Isotropic Pro Temperatures EX PRXY	A Define Material Model E Material Models Material Material Models Material Material Models Material Material Mate	S Tary e Units Tary Check and the set of the	S Alterial Model Behavior S Alterial Model Softmed Material Models Defined Target State Target Stat	Alterial Bodk, Favorite Heb Material Models Defined Material Models Defined Material Models Defined Material Models Defined Material Models Number Material Number

Fig. 15. Defining material properties

Define mesh density:

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).

ANSYS Toolbar	MeshTool	,
SAVE_DB RESUM_	Element Attributes:	
ANSYS Main Menu	Global 💌 Set	
 Preferences Preprocessor 	Smart Size	Element Sizes on Picked Lines
Element Type Real Constants	Fine 6 Coarse	[LESIZE] Element sizes on picked lines SIZE Element edge length
Material Props Sections Modeling	Size Controls:	NDIV No. of element divisions
Modeling Meshing Mesh Attribut	Global Set Clear	(NDIV is used only if SIZE is blank or zero) KYNDIV SIZE,NDIV can be changed
Element Size on Picked Li	Lines Set Clear	ANGSIZ Division arc (degrees)
Pick C Unpick	Copy Flip	(use ANGSIZ only f number of divisions (NDIV) and element edge length (SIZE) are blank or zero)
<pre>⑤ Single C Box C Polygon C Circle C Loop</pre>	Laver Set Clear Keypts Set Clear	Clear attached areas and volumes
Count = 1 Maximum = 8 Minimum = 1	Mesh: Areas	OK Apply Cancel Hep
Line No. = 7 © List of Items C Min, Max, Inc	Free C Mapped C Sweep	
, ;	Mesh Clear	
OK Apply Reset Cancel	Refine at: Elements	
Pick All Help	Refine	

Fig. 16. Setting of size control on lines

Mesh the area of the plate (free meshing).

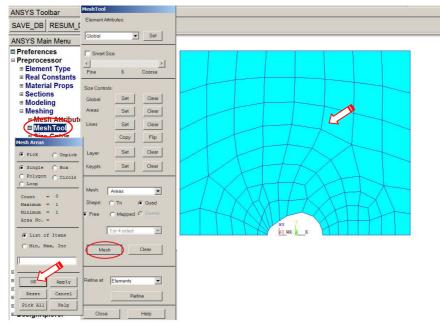


Fig. 17. FE mesh

3.2. Solution

Defining boundary conditions:

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

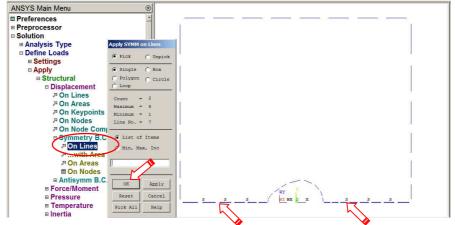


Fig. 18. Setting symmetry BC on lines

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

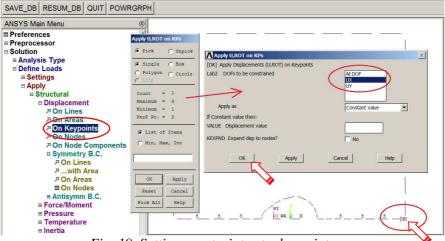


Fig. 19. Setting constraints at a keypoint

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

ANSYS Main Menu	8	T
Preferences	Apply PRES on Lines	
Preprocessor		Apply PRES on lines
Solution	Pick C Unpick	[SFL] Apply PRES on lines as a Constant value
Analysis Type	(Single (Box	
Define Loads	C Polygon C Circle	If Constant value then:
Settings	C Loop	VALUE Load PRES value -20
Apply Structural	Count = 0	If Constant value then:
	Count = 0 Maximum = 8	Optional PRES values at end J of line
Displacement Force/Moment	Minimum = 1	(leave blank for uniform PRES)
Pressure	Line No. =	(leave blank for uniform PRCS) Value
		, value
ROn Areas	(List of Items	
P On Nodes	(Min, Max, Inc	
P On Node Comport		
P On Elements		
P On Element Com		-
From Fluid Analy	OK Apply	OK Apply Cancel Help
P On Beams	Reset Cancel	
Temperature		
Inertia	Pick All Help	WY Y
Pretnsn Sectn		s s s (x s s s s
Gen Plane Strain		
Other		

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

ANSYS Main Menu	1
Preferences	
Preprocessor	/STATUS Command
Solution	Ffe
Analysis Type Define Loads	
Load Step Opt	SOLUTION OPTIONS
SE Managemer	
Results Tracki	r DEGREES OF FREEDOM UX UY
Bolve	ANALYSIS TYPE
Current LS	Solve Current Load Step
From LS File	SOLVE1 Begin Solution of Current Load Step
■ Partial Solu ■ Manual Rezoni	LOAD STEP NUMBER
Multi-field Set	TIME AT END OF THREVIEW the summary information in the lister
ADAMS Conne	
Diagnostics	PRINT OUTPUT CON
Unabridged Me	
General Postproc	
TimeHist Postpro	
Topological Opt ROM Tool	▲
DesignXplorer	S S S (WZ WX X) S S S S
Design Opt	
Prob Design	

Fig. 21. Starting solution process

3.3. General Postprocessor

Present results in the form of maps:

a) Plot directional displacements UY (in Y):

ANSYS Main Menu (8)	
Preferences	NODAL SOLUTION
Preprocessor	STEP=1 MX
Solution	50B =1
General Postproc	UY (AVG)
Data & File Opts	RSYS=0 DMX =.013079
Results Summary	
Read Results	
Failure Criteria	d
Plot Results Favorites	
Deformed Sha	
Contour Plot	
Nodal Solu	ent of displacement
Element So	ent of displacement
Elem Table Displacem	ent vector sum
ELine Elem F Stress	
Vector Plot Plot Path Item	
Concrete Plot Undisplaced shape I ThinFilm	key
	key Deformed shape only
Query Results Scale Factor	Auto Calcul • 1912.12264408
Detions for Out	
E Reculte Viewer	0056 .0084 .009799 .011199 .012599
Additional Options	(g) .007 .009799 .012599
Nodal Calcs	OK Apply Cencel Help
Flement Table	

Fig. 22. Plotting UY displacement

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

ANSYS Main Menu	(8)
Preferences Preprocessor Solution General Postproc	MODAL SOLUTION 97E0-1 1 1 1 1 1 1 1 1 1 1 1 1
Data & File Opts Results Summar	RE923=0 DMX = 013074
	Contour Nodal Solution Data
	Item to be contoured
Deformed St Contour Plot Nodal Sol Element 9 Elem Tabl Elem Elem Vector Plot Plot Path Iter	Area Notes A
	Undisplaced shape key Undisplaced shape key Deformed shape only Scale Factor Auto Calcul [1912.12264408
Options for Ou Results Viewer Write PGR File	Additional Options
■ Nodal Calcs ■ Flement Table _	UK Apply Cancel Help

Rys. 23. Plotting SY stress

c) Plot Von Mises equivalent stress SEQV.

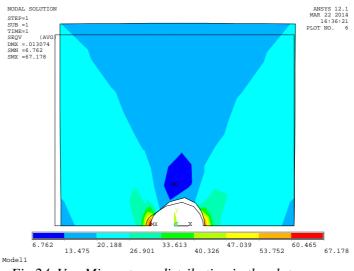


Fig.24. Von Mises stress distribution in the plate

Saving displayed plot to graphics file:

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).

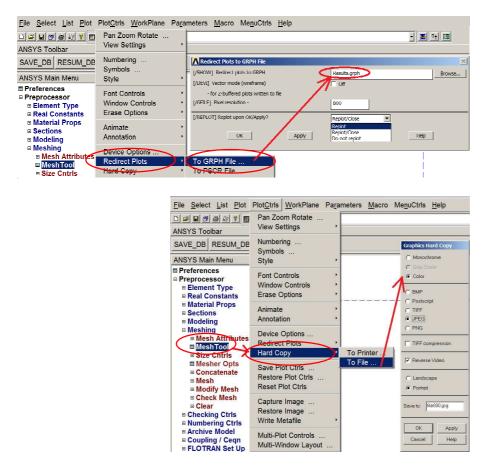


Fig. 25. Saving graphic files

<u>Diagrams are used to present stress</u> (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):

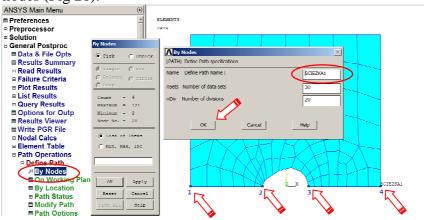


Fig. 26. Defining path and path options

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.

ANSYS Main Menu	(*)				
 ■ General Postproc ■ Data & File Opts ■ Results Summary 	Map Result Items onto Path [PDEF] Map Result Items onto Path Lab User label for item				×
 Bead Results Bead Results Plot Results ■ List Results 	Item,Comp Item to be mapped	<	DOF solution Stress Strain total Energy	Y-direction SX Y-direction SY Z direction SZ XY shear SXY	
■ Query Results ■ Options for Outp ■ Results Viewer Write PGR File	[AVPRIN] Eff NU for EQV strain		Strain-elastic Strain-thermal	YZ-shear SYZ Y direction SY	
w Nodal Calcs Element Table Path Operations	Average results across element [/PBC] Show boundary condition symbol Show path on display		Yes		—
■ Define Path □ Delete Path □ Plot Paths □ Recall Path □ Map onto Path	ОК	Лрріу	Cancel	Help	
■ Piot Path Item ■ Linearized Strs ■ List Linearized ■ Add ■ Multiply					SCIEZKAL

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).

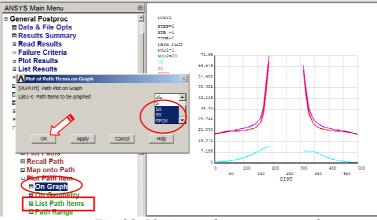


Fig. 28. Plotting path items on a graph

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} –
SX^A , SX^B –
$\alpha_{FE}^{A} = SY_{max}^{A} / \sigma_{M} -$
$\alpha_{FE}{}^B = SY_{max}{}^B / \sigma_M -$
$\sigma_M = P/(b-2a)/\delta$ –
α_T^A , α_T^B –

Maximum normal stress in Y at point A and B,
Stress in X at point A and B,
stress concentration factor at the left notche (*punkt A*),
stress concentration factor at the right notche (*punkt B*),
Mean normal stress in the symmetry plane,
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 ^₄						
SX ^B						
SEQV _{max}						
α _{FE} ^A						
α _{FE} ^B						
$\sigma_M = P/(b-2a)/\delta =$		Plots 1) FE mesh. needed → 2) UY(x,y) (should be 3) SY(x,y)			Final report: 1) Introduction 2) Assumptions for the modeling 3) model description (solid model,	
$\alpha_T^A =$		archived	4) SX(x,y)		mesh, boundary cond. and loads; 4) Results	
$\alpha_T^B =$		during program session)	5) SEQV(x,y) 6) Graph:SY(s)	,SX(s),SEQV(s)		

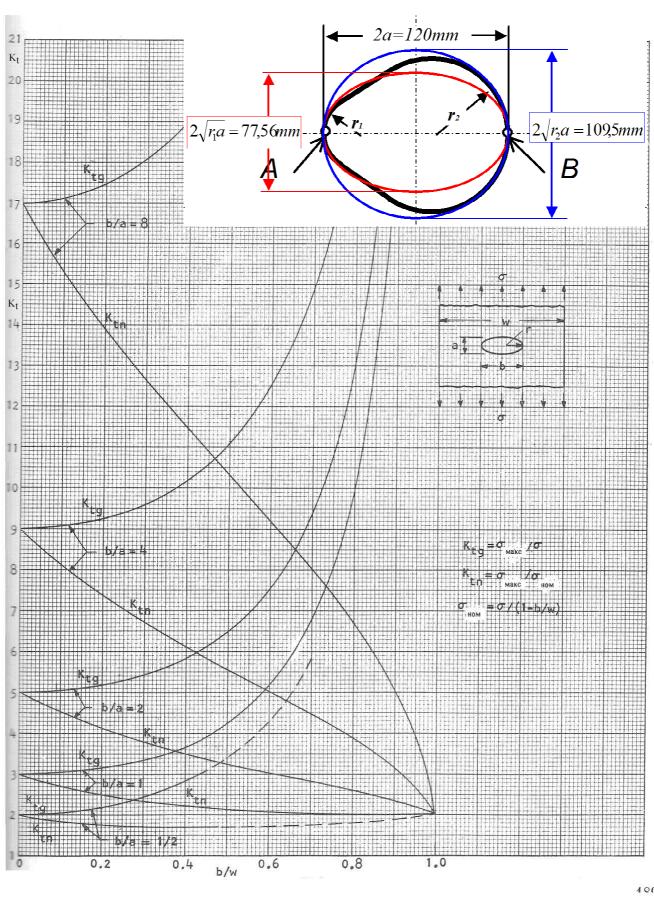


Fig. 29. Method of determining the stress concentration factor from the chart ($\alpha_T = K_{tn}$)

100