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, $E=7 \cdot 10^4 MPa$, 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 E		<u>Z-</u> +Z
ANSYS Main Menu	Offset WP to	
Preferences	Align WP with	• Snaps
Preprocessor Element Type	Change Active CS to	, Y, 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	۲	
Preferences Preprocessor Element Type Real Constants Material Props Sections	2	
Modeling		
Create B Keypoints	Circular Area by Dimensions	×
B Lines	[PCIRC] Circular Area by Dimensions	
□ Areas	RAD1 Outer radius	25
Arbitrary	RAD2 Optional inner radius	
≫ Solid Circle ≫ Annulus	THETA1 Starting angle (degrees) THETA2 Ending angle (degrees)	180
Partial Annulus	OK Apply Cancel	Help
By Dimensions ■ Polygon → Area Fillet		
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	₽⊦
	٨		

Fig.8. Plotting lines

g) Create line at angle to two lines:

ANSYS Main Menu	(8)			1
Preferences	-			6
Preprocessor				6
Element Type				6
Real Constants	At Angle to 2 Lines			e
Material Props Sections	Pick C Unpick			
	Pick OUnpick		- L	6
Modeling Create	€ Single C Box			_
Keypoints	C Polygon C Circle			
	C Loop		$\langle \rangle$	
	Count = 0		$\langle \rangle$	
A Straight Line	Maximum = 2		λ	
≫ In Active Coord	Minimum = 2			
P Overlaid on Area	Line No. =	WZ WX X		
P Tan to 2 Lines	(List of Items	A Straight Line at Angle to 2 Lines		
	C Min, Max, Inc			- 1-
Norm to 2 Lines		[L2ANG] Create a Straight Line at Angles to 2 Existing Li		
At angle to line		NL1,NL2 Existing lines	8 5	
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 by ns		C
Areas	Pick All Help	OK Apply	Cancel Help	
Volumes		лрру	- Current Trep	

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> (eight node PLANE183 or four node PLANE182) and plane stress behaviour (*Plane stress*):

ATTENTION: for Solid 182 choose "Enchanced strain"

	(1					
Main Menu	▲ Element Types			×		
Preferences						
Preprocessor						
Element Type	Defined Element T	ypes:				
Add/Edit/Delete	NONE DEFINED					
		Library of Element Types				×
Switch Elem Typ		Library of Element Types	S	Structural Mass	 Quad 4 node 182 	
Add DOF	Add	, ,,		Link	8 node 183	
Remove DOFs	Aud			Beam	Brick 8 node 185	_
Elem Tech Conti				Pipe	20node 186	
Real Constants			•	Solid Shell	concret 65	<u> </u>
Material Props	Close		L		8 node 183	
		Element type reference number		1		
Sections		Lentence of performance namber	Ľ	•		
Modeling						
Meshing		OK	Apply	Cancel	Help	
Checking Ctrle						

Fig.13. Choosing element type

Main Menu	A Element Types	×	
Preferences			
Preprocessor	Defined Element Types:	NPLANE183 element type options	×
Element Type Add/Edit/Delete	Type 1 PLANE183	Options for PLANE183, Element Type Ref. No. 1	
Switch Elem Typ		Element shape K1 Quadrilateral	
■ Add DOF ■ Remove DOFs		Element behavior K. Plane stress	
Elem Tech Conti Real Constants		Element formulation K6 Axisymmetric	
Material Props	Close	(NOTE: Mixed formulation is not with plane stress) Plane strain Plane strain Genri plane strn	
		OK 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*):



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_		
ANSYS Main Menu	Global 💌 Set	
 Preferences Preprocessor Element Type 	Smart Size	Element Sizes on Picked Lines [LESIZE] Element sizes on picked lines
Real Constants Material Props Sections	Fine 6 Coarse	SIZE Element edge length NDIV No. of element divisions 8
■ Modeling ■ Meshing ■ Mesh Attribut ■ Mesh Tool	Global Set Clear Areas Set Clear	(NDIV is used only if SIZE is blank or zero) KYNDIV SIZE,NDIV can be changed SPACE Spacing ratio
Element Size on Picked Li Pick C Unpick Single C Box	Lines Set Clear Copy Flip Laver Set Clear	ANGSIZ Division arc (degrees) (use ANGSIZ only if number of divisions (NDIV) and element edge length (SIZE) are blank or zero)
C Polygon C Circle C Loop	Keypts Set Clear	Clear attached areas and volumes No
Count = 1 Maximum = 8 Minimum = 1	Mesh: Areas	OK Apply Cancel Help
Line No. = 7 © List of Items © Min, Max, Inc	Shape: C Tri C Quad Free C Mapped C Sweep 3 or 4 sided	
	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	۲	ţ
Preferences		
Preprocessor	Apply PRES on Lines	
■ Solution	Pick O Unpick	Apply PRES on lines
	• Fick () Unpick	[SFL] Apply PRES on lines as a Constant value
Analysis Type	G Single (Box	
Define Loads	C Polygon C Circle	If Constant value then:
Settings	C Loop	VALUE Load PRES value
Apply		
Structural	Count = 0	If Constant value then:
Displacement	Maximum = 8	Optional PRES values at end J of line
Force/Moment	Minimum = 1	(leave blank for uniform PRES)
E Pressure	Line No. =	Value
On Lines		
P On Areas	List of Items	
>> On Nodes	C Min, Max, Inc	
On Node Composition		
On Elements	I	
On Element Com		
From Fluid Analy	OK Apply	OK Apply Cancel Help
P On Beams		
Temperature	Reset Cancel	
Inertia	Pick All Help	🔰 🛛 💘 🕺 🔪
Pretnsn Sectn		
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	
Preferences	
Preprocessor	/STATUS Command
Solution Fie	
Analysis Type Define Loads	
Load Step Opts	SOLUTION OPTIONS
	PROBLEM DIMENSIONALITY
	DEGREES OF FREEDOM UX UY
Borve Current LS Gond LS File Partial Solu Manual Rezoni Matui-field Set L ADAMS Connec Diagnostics Unabridged Me General Postpro TimeHist Postpro Topological Opt	ANALVSIS TYPESTATIC (STEADV-STATE) GLOBALLY ASSEMLED MATRIX SYMMETRIC [SOLVE] Begin Solution of Current Load Step LOAD STEP NUMBER TIME AT END OF TI. Review the summary information in the lister NUMBER OF SUBSTEI window (entitled "/STATUS Command"), STEP CHANGE BOUNT then press OK to start the solution. PRINT OUTPUT CON DATABASE DUTPUT
ROM Tool DesignXplorer Design Opt Prob Design	<u>s</u> <u>s</u> <u>s</u> <u>s</u> <u>x</u> <u>x</u> <u>x</u> <u>s</u> <u>s</u> <u>s</u>

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 Solution		STEP=1 SUB =1 MX
General Postproc		TIME=1 UY (AVG)
Data & File Opts		RSYS=0 DMX =.013074
Results Summary Read Results	Contour Nodal Solution Dat	ta X
Failure Criteria	Item to be contoured	
Plot Results	Barravorites	
Deformed Sha Contour Plot	Nodal Solution	
Nodal Solu	DOF Solution	
Element Go		ent of displacement
Elem Table	Displaceme	
Une Elem F Vector Plot	Stress	
Plot Path Item		
Concrete Plot	Undisplaced shape k	iey
ThinFilm List Results	Undisplaced shape k	tey Deformed shape only
Query Results	Scale Factor	Auto Calcul 1912.12264408
Options for Out		056 .0084 .011199
Results Viewer Write PGR File	Additional Options	.007 .009799 .012599
Nodal Calcs		OK Apply Cancel Help
🕫 Element Table 📃		

Fig. 22. Plotting UY displacement

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

ANSYS Main Menu	8	
Preferences	-	NODAL SOLUTION STEE=1
Preprocessor Solution		SUB =1 TIME=1
General Postproc Data & File Opts		SY (AVG) RSYS=0 DKX =.03074
Results Summary Read Results	Contour Nodal Solution Data	eun =1 217
■ Failure Criteria ■ Plot Results	Item to be contoured	
Deformed St Contour Plot Contour Plot Contour Plot Contour Plot Elem Tabl Line Elem Vector Plot Plot Path Itel Concrete Plc	Favorites Acodal Solution Conponent Y-Component Y-Y-Component Y-Y-Y-Y-Y-Y Y-Y-Y-Y-Y-Y Y-Y-Y-Y-	
ThinFilm List Results Query Results	Undisplaced shape key	Deformed shape only
Options for Ou	Scale Factor	Auto Calcul 1912.12264408
Results Viewer Write PGR File	Additional Options	B2.669 40.507 48.346 64.022 40.507 56.184 71.86
Nodal Calcs Flement Table		OK 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).



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	۲				
Data & File Opts	Map Result Items onto Path [PDEF] Map Result Items onto Path				×
Results Summary Read Results	Lab User label for item				
Failure Criteria Plot Results	Item,Comp Item to be mapped	<	DOF solution Stress Strain-total	Y-direction SX Y-direction SY Z-direction SZ	
List Results Query Results Options for Outp			Energy Strain-elastic Strain-thermal	XY-shear SXY YZ-shear SYZ Y-direction SY	<u> </u>
Results Viewer Write PGR File	[AVPRIN] Eff NU for EQV strain			Polecion 31	
Nodal Calcs Element Table	Average results across element		Yes		
Path Operations Define Path	[/PBC] Show boundary condition symbol Show path on display		□ No		
■ Delete Path ■ Plot Paths	OK	Apply	Cancel	Help	-
Recall Path Map onto Path					
 ■ Plot Path Item ■ Linearized Strs ■ List Linearized ■ Add ■ Multiply 				×	SCIEZKAI

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)
- 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 (<i>punkt A</i>),					
$\alpha_{FE}^{B} = SY_{max}^{B} / \sigma_{M} -$	stress concentration factor at the right notche (<i>punkt B</i>),					
$\sigma_M = P/(b-2a)/\delta$ –	Mean normal stress in the symmetry plane,					
$\alpha_T^A, \alpha_T^B -$	Theoretical values of stress concentration factors taken from the					
	literature.					

Discuss the results.

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



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

100