

1

Basic FEA and Savin Solution for Elliptical Holes

Notes prepared by:

Mr. Daniel Sasser MSME Candidate

April 2015

Accessing Ansys for ME Students



- <u>https://www.me.washington.edu/remotedesktop</u>
- Then follow the access instructions, using your Netid and password (same as MyUW)
- Open Ansys 16.0 (folder on desktop)
- Open Workbench or Mechanical APDL
- AA students should speak with Brian Leverson about access

Workbench Setup, Square Panel Example using ACP



- Add ACP (pre) drag and drop
 - Engineering Data add material as needed (orthotropic elasticity, etc) (units)
 - Geometry Sketch desired shape (12" square) (units)
 - Be careful with placement
 - Import CAD file
 - Create>Surface from sketch>Select sketch
 - Generate
 - Model (units)
 - Select surface body, apply thickness
 - Mesh (can be refined, mapped, etc)
 - Generate
 - Setup (units)
 - Fabric ply thickness (0.007161 in)
 - Stackup stacking sequence and order (top down or bottom up, symmetry or not)
 - Oriented Selection Set elements to apply lay-up to and which direction to stack
 - Modeling Groups number of plies and orientation (12 plies, 0 deg)

Workbench Setup, Square Panel Example using ACP



- Static Structural drag and drop on ACP (pre) through Setup
 - BC's, Loads, Output geometry (paths, probes, etc)
 - Line pressure along top (stress resultant, 100 lb/in)
 - Displacement along bottom (y = 0)
 - Outputs (normal stress, strain, deformation, etc)
 - Solve
- Compare to CLT (Professor Tuttle's code)
 - Unidirectional panel constant strain and stress
 - $\varepsilon_{11} = 46 \times 10^{-6}$ $\varepsilon_{22} = -15 \times 10^{-6}$
 - $\sigma_{11} = 1163.7 \text{ psi}$ $\sigma_{22} = -7.88 \times 10^{-7} \text{ psi}$



MAPDL Setup, Square Panel



- Open Ansys Mechanical APDL
- Double click preferences
 - Select Structural Analysis



- Select Preprocessor>Element Type>Add/Edit/Delete
 - Add type Solid>Quad 183 (adds element type PLANE 183)

A Center of Excellence



Elle Select List Plot Plot	ris <u>vv</u> orkPlane Pla <u>r</u> ameters <u>M</u> acro Me <u>n</u> uCtris	
D 🛩 🖬 🕫 🛎 🖉 📰		
Tot	Element Types X	
Defend Element Terrer		
Ma Type 1 PLANE18		
E F		ANSYS
🗆 F		R16.0
		Academic
		APE 17 2015
		18:35:10
		A PLANE 183 element type options
		Options for PLANE183, Element Type Ref. No. 1
		Flament dans K1
Add	Options	Cursic siepe to Quadristeral
		Element behavior K3 Plane strs w/thk
		Element formulation K6 Pure displacemnt
Close	Help	(NOTE: Mixed formulation is not valid with plane stress)
Solution		
General Postproc		OK Cancel Help
TimeHist Postpro BOM Tool		
Prob Design		
Radiation Opt		
Session Editor		

- Select Type 1>Options
- Change element behavior to 'Plane stress with thickness'





- Real Constants>Add/Edit/Delete>Add Thickness
 - Created a surface model, then applied thickness to it, but the model is still just a surface





- Material Props>Material Models
 - Structural>Linear>Elastic>Orthotropic
 - Set material properties

Create Geometry





- Modeling>Create>Areas>Rectangle
 - By 2 Corners at (0,0) and (12,12)



Meshing

- Meshing>Size Cntrls>ManualSize>Areas>All Areas
 - Set edge length (0.1 results in mesh seen in blue)
 - This sets the size of the mesh, the next step will create it





- Meshing>MeshTool (window on far right)
 - Select 'Mesh' button in red box
- Select the area, then click ok (window on left)



Boundary Conditions and Loads



Eile Select List El	ot Plot <u>C</u> tris <u>W</u> orkPlan	e Pa <u>r</u> ameters <u>M</u> acro	MenuCtris Help		l mul		
	7 8 📕						
Toolbar	The first second						
SAVE_DB RESUM		PH					
Main Menu	@						
Preferences		1				ANSYS	
∃ Preprocessor		ELEMENTS				R16.0	
Analysis Type				Λ	Apply LIROT on Lines		×
Define Loads							
Apply				[DL] Apply Displacement	ts (U,ROT) on Lines		
Structural	1000000 N			Lab2 DOFs to be constr	ained	All DOF	
⊟ Displace	nes					UY	
⊅ On An	eas						
P On Ke P On No	odes						
P On No	de Components						
Apply U,ROT on Lines	try B.C.						
Pick C Unpick	nent						
G Single C Box	Ire						
C Polygon C Circle							
C Loop	ectn			Apply as		Constant value	-
Count = 1	Strain			VALUE Displacement va	lue	d	
Minimum = 1	Intr						
Line No. = 4	F1				1	-1	
• List of Items				<u></u> OK	ApplyCancel	Help	
C Min, Max, Inc							
	2140)			Filmen and a statement of the statement			1
P	SWISJ						
0K Applv							
Reset Cancel							
Pick All Heln	þn						
addp							
∃ General Postproc							
ImeHist Postpro BROM Tool							
∃ Prob Design							
■ Radiation Opt	<u>↓</u>						

- Solution>Define Loads>Apply>Structural>Displacement>On Lines
 - Select left edge (highlighted in purple), click ok
 - Constrain UX = 0 (cannot move to the left or right)



- Apply>Structural>Displacement>On Nodes
 - Select lower left corner, click ok, set UY = 0
 - Will not solve unless you do this due to rigid body motion





Eile Select List Plot PlotCtris WorkPlane Parameters Macro MenuCtris Help 🔹 🛣 🤨 🛄 Toolbar SAVE_DB RESUM_DB QUIT POWRGRPH Main Menu ANSYS R16.0 Preferences ELEMENTS Preprocessor Solution Academic Analysis Type □ Define Loads APR 17 2015 Settings B Apply B Structural Displacement Force/Moment ۸ х Apply PRES on lines B Pressure ➢ On Line [SFL] Apply PRES on lines as a > On Areas Constant value • [≫] On Nodes le Components Apply PRES on Lines If Constant value then: nents @ Pick C Unpick ment Compone VALUE Load PRES value -1163.71 luid Analy (Single (Boz C Polygon C Circle ure If Constant value then: C Loop ectn Optional PRES values at end J of line Count = 1 Strain Maximum = 4 (leave blank for uniform PRES) Minimum = 1 Intr Value Line No. = 2 . List of Items C Min, Max, Inc (CMS) OK Apply Cancel Reset Pick All Help on OK Unabridged Menu Apply Cancel Help General Postproc TimeHist Postpro B ROM Tool Prob Design

- Apply>Structural>Pressure>On Lines
 - Select right edge (in purple), click ok, load is N_{xx}/thickness
 - Negative because pressure is towards the object







- Solution>Solve>Current LS, click ok
 - 'Solution is done!' Window opens, click close

Post Processing



Eile Select List Plot PlotOtris WorkP	Plane Parameters Macro Menu	Ctrls Help		T met and met			
□ ≌ ∎ @ @ @ ? 閏			-				
Toolbar							
SAVE_DB RESUM_DB QUIT POWR	GRPH						
Main Menu 🛞							
Preferences -	1					ANSYS	
Preprocessor	ELEMENTS No.	N	Contour Nodal Solution Da	ta	×	R16.0	
Solution		the sector is a sector use of				Anadomia	
General Postproc		Item to be contoured				Academic	
Data & File Opts		Revorites			-	APR 17 2015	
Results Summary		Modal Solution				12.00.07	
Read Results	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	DOE Solution				13:08:07	
Failure Criteria	1	Strocc					
Plot Results		Total Mochanical	l Otrain				
Deformed Shape			I OURIN				
Contour Plot		Accompone	nt ontoiai mechanicai sirain				
Nodal Solu		P-Component	ni oi totai mechanicai strain				
Element Solu		Z-Componer	nt of total mechanical strain		_		
Elem Table		XY Shear tot	tal mechanical strain				
Line Elem Res		YZ Shear tot	al mechanical strain				
Vector Plot		😥 XZ Shear tot	tal mechanical strain				
Plot Path Item		1st Principal	l total mechanical strain				
Concrete Plot	1	2nd Principa	al total mechanical strain				
ThinFilm		3rd Principa	l total mechanical strain				
List Results							
Query Results							
Options for Outp		- Undigelaged shape liqu					
Results Viewer		Undisplaced snape key					
Nodal Calcs		Undisplaced shape key	Deformed shape only		-		
Element Table		Carla Fonter	Late Ordenteed	10			
Path Operations		Scale Factor	Auto Calculated	٩Ľ			
Surface Operations							
E Load Case		Additional Options			8		
Check Elem Shape				[
Write Results	1		OK	Apply Cancel	Help		
ROM Operations							
Submodeling							
Safety Factor							
Define/Modify							
Nonlinear Diagnostics							
Reset							
Manual Rezoning							
TimeHist Postpro							
I ROM Tool							
Prob Design							
Radiation Ont							

 General Postproc>Plot results>Contour Plot>Nodal Solu>Total Mechanical Strain>X-component



D 📽 🖬 🔊 🎒 🌆 😵 🐒 📰 Toolbar SAVE DB RESUM DB QUIT POWRGRPH Main Menu 1 ANSYS R16.0 Preferences NODAL SOLUTION Preprocessor Solution Academic STEP=1 General Postproc SUB =1 Data & File Opts APR 17 2015 TIME=1Results Summary 13:17:39 Read Results (AVG) EPTOX ■ Failure Criteria RSYS=0 Plot Results DMX =.580E-03 Deformed Shape SMN = .460E - 04 Contour Plot SMX =.460E-04 🖩 Nodal Solu Element Solu Elem Table Line Elem Res Vector Plot B Plot Path Item Concrete Plot ThinFilm List Results Query Results Options for Outp Results Viewer Nodal Calcs Element Table Path Operations Surface Operations Load Case Check Elem Shape Write Results ROM Operations Submodeling Fatigue Safety Factor Define/Modify Nonlinear Diagnostics MAN X 🖾 Reset Manual Rezoning TimeHist Postpro B ROM Tool .460E-04 Prob Design .460E-04 B Radiation Opt Session Editor

- Strain in x-direction = 46 microstrain
- Why is it constant?



D**ĕ∎Ø**₿&?**⊠**

Toolbar

SAVE_DB RESUM_DB QUIT POWRGRPH

Main Menu 🛞	1		
Preferences	1	A	NSYS
Preprocessor	NODAL SOLUTION		R16.0
Solution	CULTED 1		
General Postproc	STEP=1		cademic
🖾 Data & File Opts	SUB =1		2015
Results Summary	TIME=1	12.1	2013
Read Results	EPTOY (AVG)	13:1	0:09
Failure Criteria	BSYS=0		
Plot Results	DMY - 590F 03		
Deformed Shape	DMA JOUE-0J		
Contour Plot	SMN =147E-04		
Modal Solu	SMX =147E-04		
Element Solu			
Elem Table			
Line Elem Res			
Vector Plot			
Plot Path Item			
Concrete Plot			
ThinFilm			
List Results			
Query Results			
Options for Outp			
Results Viewer			
Nodal Calcs			
Element Table			
Path Operations			
Surface Operations			
Load Case			
Check Elem Shape			
Write Results			
ROM Operations			
Submodeling			
Safety Factor			
Define/Modify	Y		
Nonlinear Diagnostics			
Reset	in X		
Manual Rezoning			
⊞ TimeHist Postpro			
I ROM Tool			
Prob Design			
Radiation Opt			
Session Editor			

🖸 🛃 📬 🛄

- Strain in y-direction = -15 microstrain
- Does this agree with CLT and Workbench?





🚽 🗿 🤨 📰

Toolbar

SAVE_DB RESUM_DB QUIT POWRGRPH



• Deformation in x-direction



SAVE_DB RESUM_DB QUIT POWRGRPH



💽 🚡 🤄 🗐

Deformation in y-direction

Savin Solution Ansys Workbench

Tangential stresses

20 mm hole (.787402 in)

Polar Coordinate System

 σ_{max} = 12651 psi

(0° defined by Ansys,90° defined by text)

8 ply, unidirectional, fiber direction is global y-axis (top to bottom)

Alternative CS (solution, local, fiber, global)







Stresses normal to the hole

$$\sigma_{max}$$
 = 1745.6 psi





Savin Solution, Circular Hole



 $SCF_{max} = 7.63$

Holes program predicts 7.7 for these properties





 Default mesh size from Ansys

$$-\sigma_{max} = 2457$$

- SCF = 1.4



• First mesh I used

$$-\sigma_{max} = 6393$$

$$-$$
 SCF $= 3.7$

90000 elements ~5 minutes to mesh ~5 minutes to solve

 $SCF_{max} = 7.2$

1500 elements around circumference of hole







26