University of Utah

ME EN 6510/5510
Introduction to Finite Elements
Fall 2005

2-D ELASTICITY
Using ANSYS for 2-D planar elasticity problems

Introduction - Planar elasticity problems are either plane stress or plane strain. Both categories of problems use the same elements (defaults to plane stress). In the Ansys program, plane42 is a 4 node isoparametric element with 2 degrees of freedom at each node. This element is a "workhorse" element, and is widely used. The ANSYS program permits internal degrees of freedom with incompatible modes in this element. This feature usually leads to faster convergence (i.e. better accuracy), but can be suppressed if desired. The following problem will illustrate how to use the program for planar problems. ANSYS also provides an advanced version of the plane 42 element. These are the Plane 82 elements with 8 nodes (4 at the element corners and 4 on the element midsides).

Example - Hole in a plate - The problem to be worked is shown in Fig 1. It is a typical stress concentration problem, consisting of a hole in an aluminum plate. It will be desired to calculate the maximum stress around the hole, which can then be used to calculate the stress concentration factor. Note that symmetry reduces the calculation to 1/4 of the actual plate. Using symmetry, where appropriate, is essential.

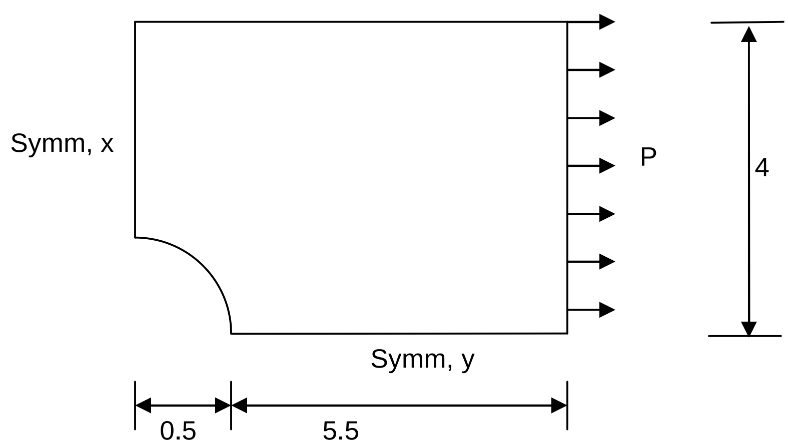

Figure 1. Hole in a flat plate problem (ANSYS model geometry).

The code required to build and solve this problem is given on the following pages. It can be typed into a prepared input file (*.inp) or entered directly into the command line. When you finish, you might want to also play with the GUI menus on your own and see if you can do the problem using that method as well.

Ansys input file	**Comments**
/title, your title	
et,1,42	! element plane42, a 4 node isoparametric planar element
mp,ex,1,10e6	! sets modulus
mp,prxy,1,.33	! sets Poisson's ratio
k,1,0.5,0	! a "key point"
k,2,4.0,0	! a "key point"
k,3,6,0	! a "key point"
k,4,6,4	! a "key point"
k,5,4,4	
k,6,0,4	! a "key point" (note that the point is at 0.0,4.0)
k,7,0,0.5	
csys,1	! changes from x,y (the default) to r, θ global cylindrical ! coordinate system. Use on-line help for other options.
k,8,0.5,45	! places key point no. 8 at r=0.5, θ=45 degrees
csys	! changes active system to the global coordinate system (default ! option for csys command (Note: csys,0 is also correct))
L,1,2,10,10	! puts a "line" with 10 segments between pts 1 & 2. The last ! segment is 10 times larger than the first. This is done to obtain ! proper mesh which is sufficiently refined at regions of high ! stress gradients and less refined at low gradient areas. It takes a ! few trials to get the right mesh depending on the complexity of ! the given problem.
L,8,5,10,10	!
L,7,6,10,10	!
csys,1	! cyl. coord, so that the lines below match the hole
L,1,8,5	! 5 segments from 1 to 8, equally spaced (the default)
L,8,7,5	! 5 segments from 8 to 7, equally spaced (the default)
csys	! changes active system to the global coordinate system
L,2,5,5	
L,5,6,5	
L,2,3,4	
L,3,4,5	
L,4,5,4	
a,1,2,5,8	! creates an area with key pts as corners (order is important)
a,8,5,6,7	! creates an area with key pts as corners
a,2,3,4,5	
amesh,all	! This creates the element mesh
nsel,s,loc,x,0,0	! select nodes on y axis (x=0)
dsym,symm,x	! define a symmetry b.c. for these nodes (same as constraining ! 'ux' displacements using 'd' command.
nsel,s,loc,y,0,0	! select nodes on x axis (y=0) (from all nodes)
dsym,symm,y	! define a symmetry b.c. for these nodes

```
nsel,s,loc,x,6,6        ! select nodes  on right side of plate  (from all nodes)
sf,all,press,-1000      ! apply tension of P=1000 units to right side of plate
nsel,all                ! reselect all nodes
/psf,press,2            ! show pressure as arrows on plots
/pbc,u,1                ! show disp. constraints  on plots
/rep                    ! replot (similar to refresh screen)
```

Once the input file has been prepared as above, check to see that it is ok by typing in the following commands to ANSYS.

```
/prep7                  !the input phase of ansys
/show, x11              !necessary if you are going to do plots
/input,filename.inp     !Filename.inp  is the name  of your input file (8 letters max for
                        !filename).  All of the commands in the input file will be
                        !executed.
```

The following commands are then used to examine your model to see that it is correct and what you want.

```
klist                   ! lists all keypoints
lplot                   ! gives a plot of the line segments
aplot                   ! plots the area.  The divisions are not elements
eplot                   ! gives a plot of elements  (a good diagnostic  tool)
/pbc,u,1                ! 1 shows the b.c.'s in plots, 0 turns this feature off
/pnum,node,1            ! 1 shows the node  no's in plots, 0 turns this off
nplot                   ! plots the nodes
```

If the model is not correct, go to the editor and fix the input file, and re-enter it as above. If correct, proceed as follows.

```
wsort,y                 ! used to minimize  bandwidth, good for larger problems.  For
                        ! large problems you need to try a sort in each direction, ansys
                        ! will  use the smallest bandwidth.
finish                  ! exits from prep7

/solu                   ! initiates solution phase
solve                   ! runs the solution
finish                  ! after solution, exits from solution phase

/post1                  ! this enters the post processing  phase, to examine answers
/show,x11               ! for x windows display
set                     ! required directive
/edge,1,1               ! Plots model edges (optional)
pldisp,1                ! plots the original  and deformed  mesh, an important  diagnostic
plnsol,s,x              ! plots a contour plot of stresses in the x direction.  Other options
                        ! sx,sy,sxy.  Good diagnostic.
plnsol,s,1              ! plots a contour  plot of sig1 principal stress
```

```
csys,1               ! change back to polar coordinates
nsel,s,loc,x,0,2     ! selects only nodes within a radius of 2 from the origin
                     ! (restore all nodes with “nsel,all”)
esln,s               ! select all elements attached to selected nodes
csys
plnsol,s,1           ! contour  plot near the region of high stresses
/pnum,node,1         ! 1 shows the node no’s in plots, 0 turns this off
nplot                ! Plots the node numbers.  Note which numbers are along the
                     ! cross-section where the stress is highest
/edge,1,0            ! turns the edge  feature off.  Necessary for the next plot
lpath,72,78          ! defines a path for detailed  plotting,  based on node numbers
pdef,sig1,s,1        ! labels  the principal  stress along  the defined  path as sig1
plpath,sig1          ! plots sig1 along the path  from node 72 to node 78.  The
                     ! numbers were obtained  from the node plot
prpath,sig1          ! prints the sig1 values, if you want to see them
finish               ! exits from post1
```

When you are done, type

```
/eof                 ! exits from Ansys
```