! Plane Stress tension of a plate with a hole
! ANSYS Rel.2020 R1.

1. Project title

Utility Menu>File>Change Title

1.1. Print “Plane Stress tension of a plate with a hole”
1.2. **OK**

2. Enter scalar parameters for geometrical sizes, material modules and pressure

Utility Menu>Parameters>Scalar Parameters

2.1. Enter scalar parameters (press “Accept” after entering each parameter, all ANSYS commands and parameter names are NOT case sensitive!)

```
A=5         ! length of the plate quarter
B=2         ! width of the plate quarter
R=0.25      ! radius of the hole
H=0.1       ! thickness of the plate
P=1e3       ! magnitude of tensile load(kgf/cm^2) 1kG = 9.8 N/m^2
```

2.2. **Close**

3. Set material properties

3.1. **Main Menu>Preprocessor>Material Props>Material Models**
(window «Define Material Model Behavior» will appear)
3.2. In window «Material Models Available» tick: **Structural**, **Linear**, **Elastic**, **Isotropic**.
3.3. For Young’s modulus EX enter *2e6*
3.4. For Poisson’s ratio PRXY enter *0.3*
3.4. **OK**
3.5. Close window “Define Material Model Behavior” choosing **Material>Exit** or **x**.
3.6. Close tab “Material Props”

4. Set finite element type

4.1. **Main Menu>Preprocessor>Element Type>Add/Edit/Delete**
4.2. Choose “Add”
4.3. Scroll down the list of elements and choose “Structural Solid”. From the right-hand list choose “Quad 8node 183” (quadrilateral eight-node finite element PLANE183 will be chosen)
4.4. **OK**
4.5. **Close**
4.6. Closes window “Element Type”

5. Build geometrical model

5.1. Define keypoints
Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS
In window “Create Keypoints in Active Coordinate System” set keypoints

NPT→ 1
X,Y → 0,0
Apply

NPT→ 2
X,Y → a,0
Apply
NPT→ 3
X,Y → a,b
Apply
NPT→ 4
X,Y → 0,b
OK
x

5.2. Define rectangular area by connecting keypoints
Main Menu>Preprocessor>Modeling >Create>Areas>Arbitrary>Through Kps
Window “Create Area thru KPs” will appear on the left
Using left button of the mouse pick all keypoints in ascending order of keypoint numbers
OK
x

5.3. Define the cirle
Main Menu>Preprocessor>Modeling >Create>Areas>Circle >Solid Circle
In window “Solid Circular Area” set radius (the circle will be created with the center in (0,0))
Radius→ *r*
OK
x

5.4. Define new area (area 3) by subtracting circle (area 2) from rectangle (area 1)
Main Menu>Preprocessor>Modeling >Operate>Booleans>Subtract>Areas
Window “Subtract Areas” will appear on the left
In graphical window pick rectangular area (area 1) → **OK**
Then pick the circle (area 2) → **OK** → **x** → **x**
The resulting area will have number 3, areas number 1 and 2 will be deleted.

6. Build finite element mesh

6.1 Plot areas and show keypoint numbers
Utility Menu>PlotCtrls>Numbering
In window “Plot Numbering Controls” tick keypoint numbering
KP Keypoint numbers→ ∨
OK
Utility Menu>Plot>Areas

6.1a. If you cannot see keypoint numbers near the hole of the plate, opt for not showing the coordinate triad:
Отмена показа глобальной ситемы координат
Utility Menu>PlotCtrls>Window Controls>Window Options
В появившемся окне «**Window Options**» выбрать
[/TRIAD] Location of triad → Not shown
OK

6.2. Set element edge length for elements around keypoints
First, set global size of element edge length for all keypoints
Main Menu>Preprocessor>Meshing>Size Cntrls>Manual Size>Keypoints>All KPs

In window “Element Sizes on All Selected Keypionts” set the size (length of the element edge) for all keypoints

SIZE Element Edge length → *B/4*

Main Menu>Preprocessor>Meshing>Size Cntrls>Manual Size>Keypoints >Picked KPs
Window “Elem Size at Picked KP” will appear on the left
Pick keypoints 5 and 6 (the ends of the arc)

Apply

In window “Element Sizes at Picked Keypoints” set smaller size of elements in order to insure mesh condensation near the hole

SIZE Element Edge length → *R/5*
OK
x

6.3. Build finite element mesh for area 3
Main Menu>Preprocessor>Meshing>Mesh>-Areas>Free
In graphical window pick the area
In window “Mesh Areas” press **OK**
x

7. Exit Presprocessor
Close Preprocessor tab

8. Enter Solution processor
Main Menu>Solution

9. Define analysis type
Main Menu>Solution>Analysis Type>New Analysis
OK (Static analysis is default)

10. Set boundary conditions

10.1. Define pressure at the right boundary of the plate
Main Menu> Solution >Define Loads>Apply>Structural>Pressure>On Nodes
In window “Apply PRES on Nodes” select “Box”
Using Box frame starting from the upper right node of the model pick all nodes of the right edge (nodes with coordinates x=a)
OK
In window “Apply PRES on Nodes” define the value of the pressure

VALUE Load PRES value → *-p*
OK
x

10.2. Set symmetry boundary conditions

First, plot lines and show line numbers
6.1 Plot areas and show keypoint numbers
Utility Menu>PlotCtrls>Numbering
In window “Plot Numbering Controls” tick line numbering

Line numbers→ ∨
OK

Utility Menu>Plot>Lines

Main Menu> Solution > Define Loads>Apply>Structural >Displacement>Symmetry B.C.>On Lines
Window “Apply SYMM on Lines” will appear
Using mouse, pick lines 9 and 10 (lines with x=0 and y=0)
OK
x x
Warning about imposing both solid and finite element boundary conditions will appear. This warning can be neglected here. Close window.

11. Solve the problem

Main Menu>Solution>Solve>Current LS
OK → Close → Close
x

12. Exit from ANSYS
Utility MenuFile>Exit
or
ANSYS Toolbar>QUIT
In Exit from ANSYS menu choose Save everything (for working with database of results later) or Quit – No Save
OK