

### ANSYS Example #3: Plane Stress Analysis of a Keyhole Specimen

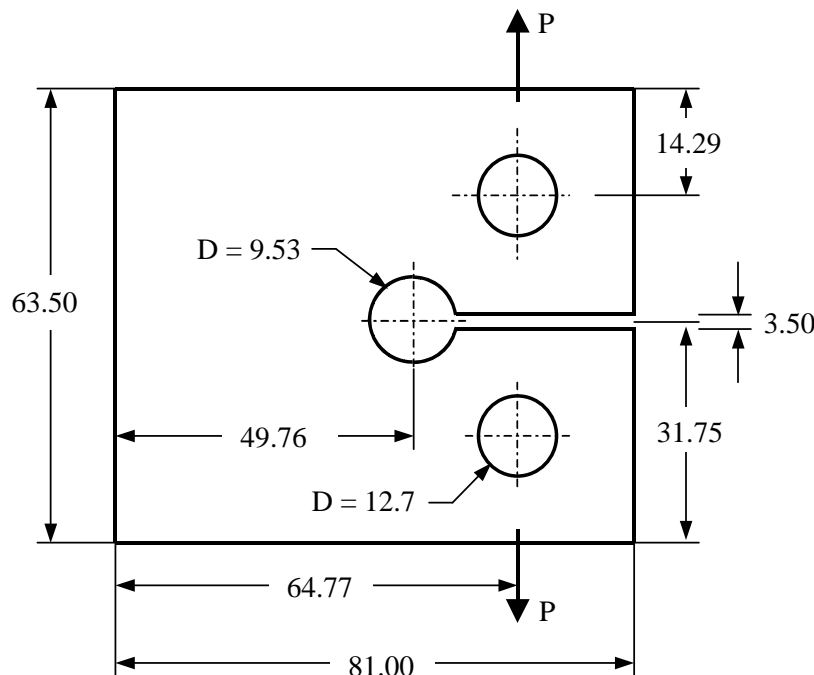
The “keyhole” specimen shown below has been used in fatigue research to evaluate the accuracy of fatigue life prediction models (note all dimensions are in mm). A cyclic load is applied to the specimen in the vertical direction until a fatigue crack propagates out from the ‘keyhole’ in the center and causes the specimen to fracture. The load ( $P$ ) is applied to the specimen by pins that are placed through the two grip holes on the right side of the specimen. Before conducting a test, however, the elastic stress concentration factor  $K_t$  due to the keyhole must be determined. For this analysis, we will assume the specimen is made of aluminum ( $E = 71$  GPa and  $\nu = 0.33$ ) with a yield strength of  $S_y = 290$  MPa and is 8.2 mm thick.

In this example, we will use ANSYS to analyze the stresses and deflections in the specimen under a static load of  $P = 3500$  N. A linear-elastic, static analysis will be performed using 8 node, 2D isoparametric plane stress elements (PLANE 82). Specifically, we wish to determine the magnitude of the stress concentration factor due to the keyhole in the center of the specimen, and the magnitude of the opening at the tip of the slot on the right edge of the specimen. The stress concentration factor is defined as the ratio of the maximum stress at the edge of the hole to the nominal stress at that point. For this specimen, the nominal stress is due to a combination of axial loading and bending across the reduced cross-section through the keyhole:

$$K_t = \frac{\sigma_{\max}}{\sigma_{\text{nom}}} \quad \text{where} \quad \sigma_{\text{nom}} = \frac{P}{A} + \frac{Mc}{I}$$

where  $A$  is the net-section area through the keyhole,  $M$  is the moment due to  $P$  about the neutral axis of the net-section area,  $c$  is the distance from the neutral axis to the edge of the hole, and  $I$  is the moment of inertia of the net-section. Using the dimensions shown below:

$$\sigma_{\text{nom}} = \frac{3500}{(0.045)(0.0082)} + \frac{(3500)(0.0423)(0.0225)}{(0.0082)(0.045)^3 / 12} = 63.0 \text{ MPa}$$



Since the geometry, loads and constraints for this specimen are symmetric, only half of the specimen needs to be analyzed. In this example, we will model the upper half of the specimen.

### ANSYS Analysis:

*Start ANSYS Product Launcher, and define Job Name as 'Keyhole\_Specimen\_1'. Then define Title and Preferences.*

**Utility Menu** → **File** → **Change Title...** → Enter 'Plane Stress Analysis of a Keyhole Specimen' → OK

**ANSYS Main Menu** → **Preferences** → Preferences for GUI Filtering → Select 'Structural' and 'h-method' → OK

### Enter the Preprocessor to define the model geometry:

*Define Element Type, and select Plane Stress option with Thickness input.*

**ANSYS Main Menu** → **Preprocessor** → **Element Type** → **Add/Edit/Delete** → Add... → Structural Solid Quad 8 node 82 (PLANE82) (define 'Element type reference number' as 1) → OK → Click Options... → Select 'Plane strs w/thk' for K3 (Element behavior) → OK → Close

*Define Real Constants (thickness of specimen) and Material Properties.*

**ANSYS Main Menu** → **Preprocessor** → **Real Constants** → **Add/Edit/Delete** → Add... → Click OK (Type 1 PLANE82) → Enter 1 for 'Real Constant Set No.' → Enter 0.0082 for THK (Thickness) → OK → Close

**ANSYS Main Menu** → **Preprocessor** → **Material Props** → Material Models → Double Click Structural → Linear → Elastic → Isotropic → Enter 71e9 for EX and 0.33 for PRXY → Click OK → Click Exit (under 'Material')

\*\*\*\*\*

There are a variety of ways to create the specimen geometry. We could define Keypoints by spatial coordinates, then define Lines between the Keypoints, and finally define Areas between the Lines (similar to what we did in the previous example). Alternatively, we can create the geometry by defining areas directly in terms of basic shapes, such as rectangles, circles, etc. In this example, we will use the latter method, as it is generally faster for complex shapes.

\*\*\*\*\*

*We will place the origin at the lower left corner (of the upper half of the specimen). First define two rectangles. The first rectangle represents the entire upper half of the specimen. The second represents the small rectangular strip that must be subtracted along the lower right corner, leading in to the keyhole.*

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Create** → **Areas** → **Rectangle** → By Dimensions → Enter 0 and 0.081 for X-coordinates and 0 and 0.03175 for Y-coordinates → Click Apply → Enter 0.04976 and 0.081 for X-coordinates and 0 and 0.00175 for Y-coordinates → OK  
**Utility Menu** → **PlotCtrls** → **Numbering...** → Click 'Area numbers' On → OK

*Now "subtract" the second Area from the first (called a "Boolean" operation).*

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Operate** → **Booleans** → **Subtract** → Areas → Select (with the mouse) Area 1 (bigger rectangle) → OK → Select (with the mouse) Area 2 (smaller rectangle) → OK

Define the two circular Areas (pin hole and keyhole), and subtract these areas from the model.

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Create** → **Areas** → **Circle** → Solid Circle → Enter 0.04976 for WP X (WP stands for Working Plane), 0 for WP Y, and 0.004765 for Radius → Apply → Enter 0.06477 for WP X, 0.01746 for WP Y, and 0.00635 for Radius → OK

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Operate** → **Booleans** → **Subtract** → Areas → Select (with the mouse) Area 3 (rectangle) → OK → Select Areas 1 and 2 (circles) → OK

\*\*\*\*\*

In this model, we anticipate that high stresses and stress gradients will occur in the vicinity of the keyhole. Consequently, when meshing the specimen, we will try to place a finer mesh around the keyhole to improve the accuracy there, and a coarser mesh away from the keyhole to reduce the size of the model. To accomplish this, we will define an additional area around the keyhole, by splitting the current area into two new areas.

\*\*\*\*\*

Define a rectangular Area around the keyhole, split the Areas using the 'Overlap-Boolean' function, and then delete the additional Area around the keyhole that is also created.

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Create** → **Areas** → **Rectangle** → By Dimensions → Enter 0.032 and 0.062 for X-coordinates and 0 and 0.01 for Y-coordinates → OK

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Operate** → **Booleans** → **Overlap** → Areas → Select 'Pick All'

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Delete** → Area and Below → Select (with the mouse) Area 2 (area that does not belong to the specimen) → OK

The Areas can now be renumbered, and Line numbers displayed.

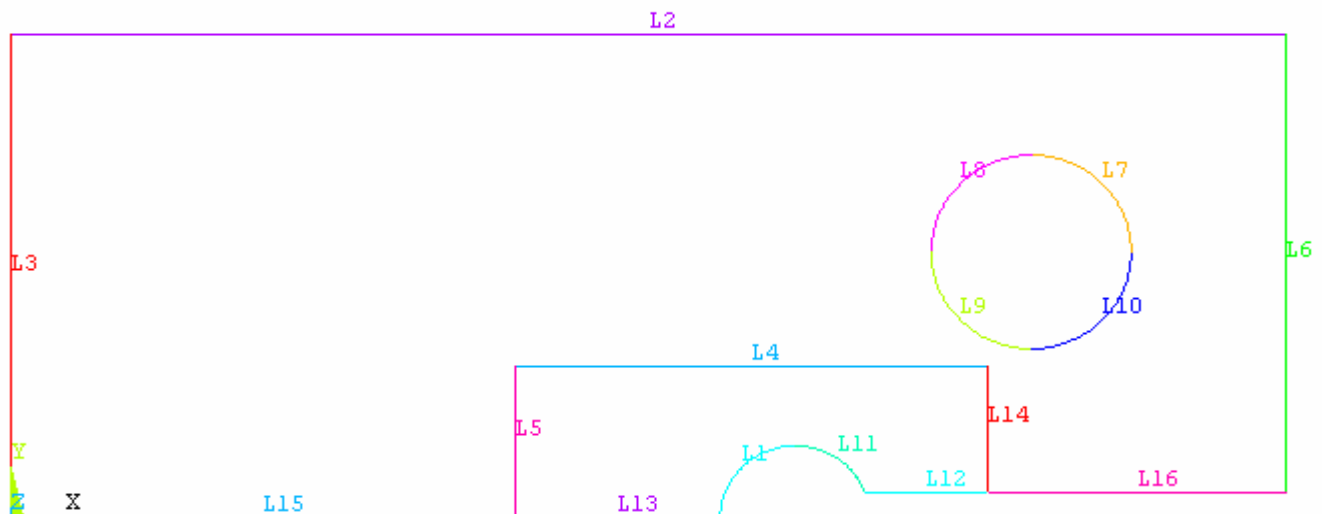
**ANSYS Main Menu** → **Preprocessor** → **Numbering Ctrl**s → **Compress Numbers** → Select All → OK

**Utility Menu** → **Plot** → **Replot** (Area numbers change to 1 and 2)

**Utility Menu** → **Plot** → **Lines**

**Utility Menu** → **Plot Ctrl**s → **Numbering...** → Click 'Line numbers' On → OK

The Areas and Lines should appear as shown below:



\*\*\*\*\*

The areas can be meshed using one of two methods:

1. **Free Meshing.** A different global element size (edge length) can be defined for each area: a big element size for the larger area and a much smaller element size for the area surrounding the keyhole, to produce a finer mesh in this region. Using the free-mesh option, ANSYS will fit elements into each area automatically, keeping each element in the area approximately the same size (corresponding to the defined element size). Note that ANSYS may place triangular elements in some locations, even though quadrilateral elements were defined.
2. **Mapped Meshing.** The mesh in the area surrounding the keyhole can be better controlled using the mapped-mesh option. To use this option, the area is divided again, and some of the lines in each area are concatenated so that the areas have a regular geometry (bounded by four lines). Some of these lines are then defined to have a certain number of divisions (corresponding to a desired number of elements), and then the area is meshed. Note that a “spacing ratio” can also be defined to bias the elements to one side of the line; i.e., create elements that are smaller near the keyhole, and gradually get larger away from the hole.

Each method will be demonstrated in this example.

\*\*\*\*\*

*Save the current model geometry in a separate database so it can be retrieved later for alternate meshing techniques.*

**Utility Menu → File → Save as... → Keyhole\_Geometry.db**

### **Free-Meshing Technique:**

*Define a Global Element Size (edge length) for each Area and mesh the Area using the Free-Mesh option, with Quad Elements. Note that this is the only option that can be used when the Area is irregular; i.e., not bounded by four sides (Lines).*

**ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Under ‘Size Controls: Global’ click Set → Enter 0.001 for ‘Element edge length’ → OK → Under ‘Mesh:’ select Areas, Quad and Free → Click Mesh → Select (with the mouse) Area 1 (area surrounding the keyhole) → OK**

**Utility Menu → Plot → Areas**

**ANSYS Main Menu → Preprocessor → Meshing → MeshTool → Under ‘Size Controls: Global’ click Set → Enter 0.003 for ‘Element edge length’ → OK → Under ‘Mesh:’ select Areas, Quad and Free → Click Mesh → Select (with the mouse) Area 2 (larger area) → OK**

### **Enter the Solution Menu to define boundary conditions and loads and run the analysis:**

*Define the Analysis Type.*

**ANSYS Main Menu → Solution → Analysis Type → New Analysis → Select Static → Click OK**

*Apply Displacement Constraints to the model. In this case, Symmetry Boundary Conditions must be applied along the lower edge of the model, Lines 13 and 15 (this restricts vertical displacements of nodes along this edge). In addition, horizontal (rigid-body) motion of the model must be prevented – we will constrain the node in the lower left corner of the model.*

**ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → Symmetry B.C. → On Lines → Select (with the mouse) Lines 13 and 15 → OK**

**ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Displacement → On Nodes → Select (with the mouse) the node in the bottom left corner (X = 0, Y = 0) → OK → Select ‘All DOF’ for ‘DOFs to be constrained’ → OK**

\*\*\*\*\*

The load (P) can be applied by different methods. The simplest method would be to apply a single concentrated force to the node at the top of the pin hole; however, this will create very high stresses (unrealistic) in the vicinity of the force. An alternative method is to apply a pressure along Lines 7 and 8 (upper edge of the pin hole). This produces a much more realistic type of loading and minimizes the error caused by the concentrated force near the point of load application. Note that both methods should produce similar results for stresses and displacements away from the pin hole, however. We can thank St. Venant for recognizing this.

In this example, we will apply a uniform pressure along the upper edge of the pin hole. The magnitude of the pressure is based on the projected area (bearing area) of the hole (not the surface area):

$$p = \frac{P}{D * t}$$

where P is the applied load, D is the hole diameter, and t is the thickness of the specimen. Using D = 12.7 mm, t = 8.2 mm, and P = 3500 N, we get p = 33.6086E6 Pa

\*\*\*\*\*

*Apply a Pressure Load to the Lines along the upper edge of the pin hole.*

**ANSYS Main Menu** → **Solution** → **Define Loads** → **Apply** → **Structural** → **Pressure** → On Lines → Select (with the mouse) Lines 7 and 8 (*top two lines along pin hole*) → OK → Enter 33.6086e6 for 'Load PRES value' → OK

*Save the Database and initiate the Solution using the current Load Step (LS).*

**ANSYS Toolbar** → SAVE\_DB (*this saves the database under the current jobname, Keyhole\_Specimen\_1*)

**ANSYS Main Menu** → **Solution** → **Solve** → Current LS → OK → Close the information window when solution is done → Close the /STATUS Command window

### **Enter the General Postprocessor to examine the results:**

*The deformed shape should be checked first.*

**ANSYS Main Menu** → **General Postproc** → **Plot Results** → Deformed Shape → Select Def + undeformed → OK

*A Contour Plot of any stress component can be generated. A Contour Plot shows lines of constant stress for the selected component; e.g.,  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\sigma_1$ ,  $\sigma_2$ ,  $\sigma_{eqv}$  (von Mises stress), stress intensity ( $\sigma_1$ - $\sigma_3$ ), etc.*

**ANSYS Main Menu** → **General Postproc** → **Plot Results** → **Contour Plot** → Nodal Solution → Select 'Stress' and 'von Mises stress' → OK

**ANSYS Main Menu** → **General Postproc** → **Plot Results** → **Contour Plot** → Nodal Solution → Select 'Stress' and 'Y-Component of stress' → OK

*The Nodal values of the Component or Principal Stresses can also be listed. The maximum and minimum value of each component is printed at the end of the listing. These stresses can be printed to a text file, or simply copied and pasted into an Excel file for additional manipulations.*

**ANSYS Main Menu** → **General Postproc** → **List Results** → Nodal Solution → Select 'Stress' and any component of stress (all stress components will be listed) → OK

**ANSYS Main Menu** → **General Postproc** → **List Results** → Nodal Solution → Select 'Stress' and any principle stress (all principle stresses, von Mises stress, and stress intensity will be listed) → OK

To obtain the magnitude of the tip opening on the right edge of the slot, list the Nodal Displacements. It will first be necessary to determine the number of the Node at this location. This can be obtained by zooming in on a Node Plot and displaying Node numbers using the “Numbering” feature under **PlotCtrls**.

**ANSYS Main Menu** → **General Postproc** → **List Results** → Nodal Solution → Select ‘DOF Solution’ and ‘Displacement vector sum’ (or just ‘Y-Component of displacement’) → OK

Alternatively, we can use the **Query Results** command to get the Nodal Solution at a particular Node. First, the PowerGraphics feature must be turned off.

**ANSYS Toolbar** → POWRGRPH → Click PowerGraphics Off → OK

**ANSYS Main Menu** → **General Postproc** → **Query Results** → Nodal Solu → Select ‘DOF solution’ and ‘Translation UY’ → OK → Select (with the mouse) the desired Node

To check the discretization error, the specimen should be meshed once more with a finer mesh, and the analysis performed again. If similar results are obtained with the two meshes, this would indicate that the discretization error is small.

### **Mapped-Meshing Technique:**

Retrieve the Model Geometry (without the mesh) by Resuming from the Geometry Database. Then change the Jobname for this second analysis.

**Utility Menu** → **File** → **Resume from...** → Select ‘Keyhole\_Geometry.db’ → OK

**Utility Menu** → **File** → **Change Jobname...** → Enter ‘Keyhole\_Specimen\_2’ → OK

To employ the Mapped Meshing technique, we will split the Area around the keyhole into two separate Areas. This will allow us to define the regular geometry necessary for Mapped Meshing. The Area will be split by defining an additional rectangular Area near the keyhole, splitting the Areas by using the Boolean-Overlap function, and then deleting the additional Area that is not part of the model.

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Create** → **Areas** → **Rectangle** → By Dimensions → Enter 0.04976 and 0.062 for X-coordinates and 0 and 0.01 for Y-coordinates → OK

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Operate** → **Booleans** → **Overlap** → Areas → Select ‘Pick All’

**ANSYS Main Menu** → **Preprocessor** → **Modeling** → **Delete** → Area and Below → Select (with the mouse) Area 6 (area that does not belong to the specimen) → OK

Compress the Area numbers and replot the Lines.

**ANSYS Main Menu** → **Preprocessor** → **NumberingCtrls** → **Compress Numbers** → Select All → OK

**Utility Menu** → **Plot** → **Replot** (Area numbers change to 1, 2 and 3)

**Utility Menu** → **Plot** → **Lines**

**Utility Menu** → **PlotCtrls** → **Numbering...** → Click ‘Line numbers’ On → OK

For the Mapped Meshing technique, a regular geometry is required (Areas bounded by four Lines only). To make the geometry regular, we must combine Lines 13 and 18 into a single Line and combine Lines 4 and 17 into another single Line. This operation is called “Concatenate.”

**ANSYS Main Menu** → **Preprocessor** → **Meshing** → **Concatenate** → Lines → Select (with the mouse) L13 and L18 → Apply → Select L4 and L17 → OK

*Set the desired number of Divisions (Elements) for each Line. A Spacing Ratio can be defined to bias the elements toward one side.*

**ANSYS Main Menu → Preprocessor → Meshing → MeshTool →** Under ‘Size Controls: Lines’ click Set → Select (with the mouse) L10 → Apply → Enter 8 for NDIV → Apply → Select L1 → Apply → Enter 12 for NDIV → Apply → Select L11, L16 and L12 → Apply → Enter 10 for NDIV and 0.4 for ‘Spacing ratio’ → OK

*Check the Spacing Ratio. We want smaller elements near the hole and larger elements away from the hole. If this is not what occurs, flip the Spacing Ratio. In our case, the Spacing Ratio on Lines L11 and L16 is opposite of what we want, so flip the Spacing Ratio.*

**ANSYS Main Menu → Preprocessor → Meshing → MeshTool →** Under ‘Size Controls: Lines’ click Flip → Select L11 and L16 → OK

*Mesh the Areas around the keyhole using the Mapped Meshing option (Quad Elements). The larger Area can still be meshed using the Free Mesh option with a defined Global Element size.*

**ANSYS Main Menu → Preprocessor → Meshing → MeshTool →** Under ‘Mesh:’ select Areas, Quad, and Mapped → Click Mesh → Select Areas 1 and 2 (*areas surrounding the keyhole*) → OK

**ANSYS Main Menu → Preprocessor → Meshing → MeshTool →** Under ‘Size Controls: Global’ click Set → Enter 0.002 for ‘Element edge length’ → OK → Under ‘Mesh:’ select Areas, Quad, and Free → Click Mesh → Select Area 3 (*larger area*) → OK

*Define Boundary Conditions and Loads in the Solution menu, and re-run the analysis. Enter the Postprocessor to view the results, as before. Again, the analysis should be re-run with a finer mesh to check for discretization error.*