Stress Concentration in Flat Plates With Circular Holes

Blake Logan
Jackson Fry
3/28/2023

To analyze the effects of stress concentration, ANSYS finite element analysis was performed on several flat plates with different hole diameters. The hole diameters were 1 in, 0.75 in, 0.5 in, and 0.25 in. Each plate was subjected to an axial tensile force of 1000 lbf. To begin the analysis, a sketch of the plate was created in the X-Y plane. The rectangular section was dimension and constrained with symmetry. A circular sketch with a 1 in diameter was constrained with symmetry to the origin. The plate will be analyzed in 2-D, therefore, a surface from sketch was created with a thickness of 0.118 in. Using the symmetry tool, the sketch was cut about its planes of symmetry. This is much more efficient as only one quarter of the plate will be analyzed. Refer to Figure 1 below.

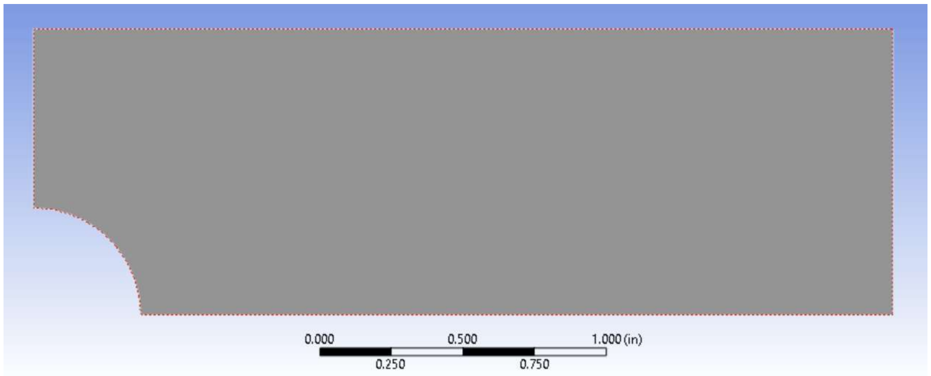

Figure 1: 2D Sketch with Symmetry

Next, the geometry was added to a static structural file. In Mechanical, a force of 500 lbf was added to the end of the element. This is because only half of the cross section is being analyzed. A normal stress solution was added with a convergence object to ensure ANSYS converges to a solution. In addition, a surface was created in the Y-Z plane and normal stress was evaluated at this surface. This allowed for the normal stress data to be exported and the stress variation to plotted in MATLAB. This process was repeated for the remaining hole diameters.

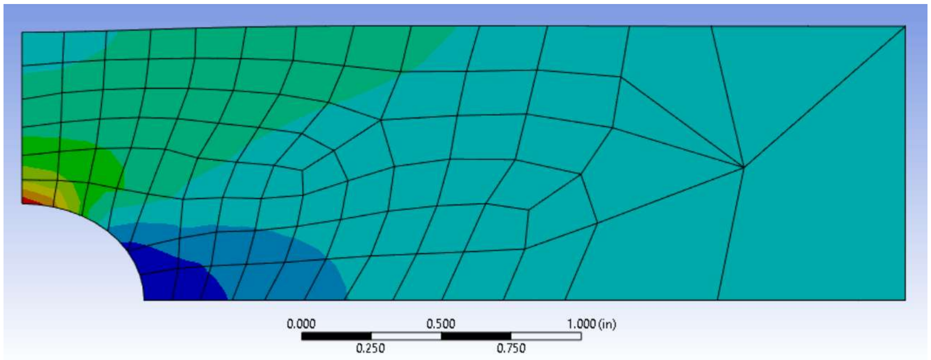

Figure 2: Normal Stress Distribution

The results of the ANSYS finite element analysis are shown below. A distribution of the normal stress along the y-axis along with the convergence plot is shown for each hole diameter. The convergence criteria for ANSYS to arrive at a solution was for the solution to be within 1% of the previous iteration.

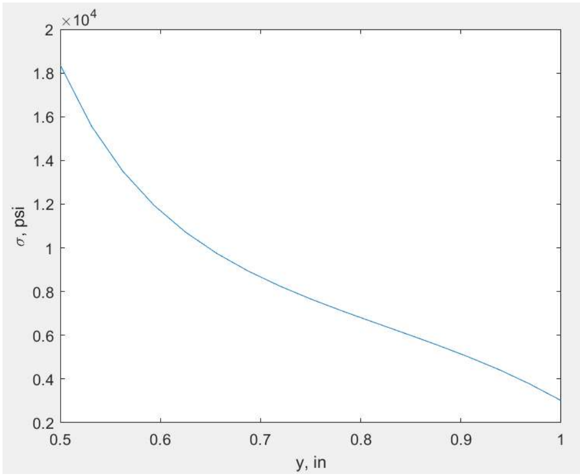

Figure 5: Stress Distribution, 1 in

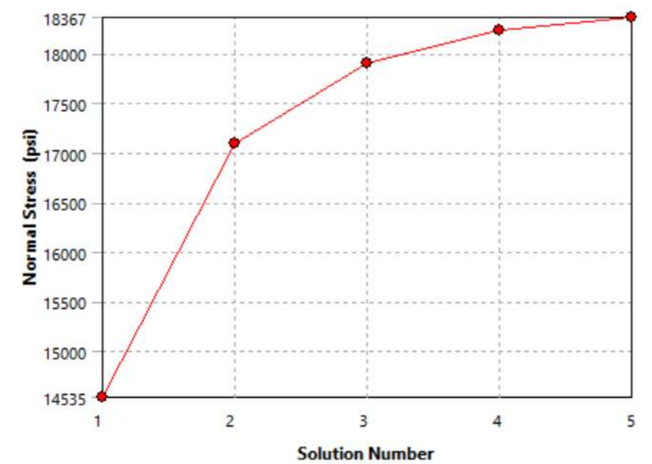

Figure 6: Convergence Plot, 1 in

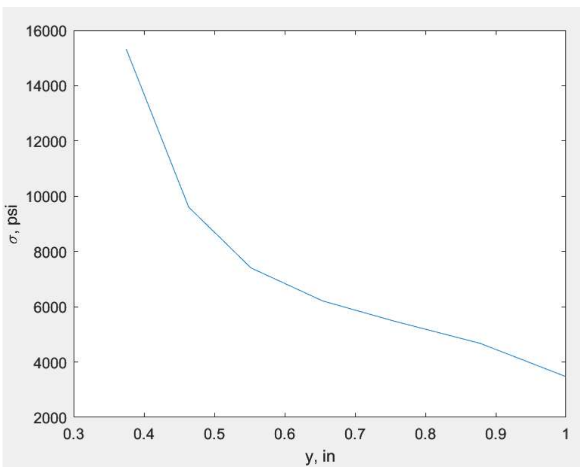

Figure 7: Stress Distribution, 0.75 in

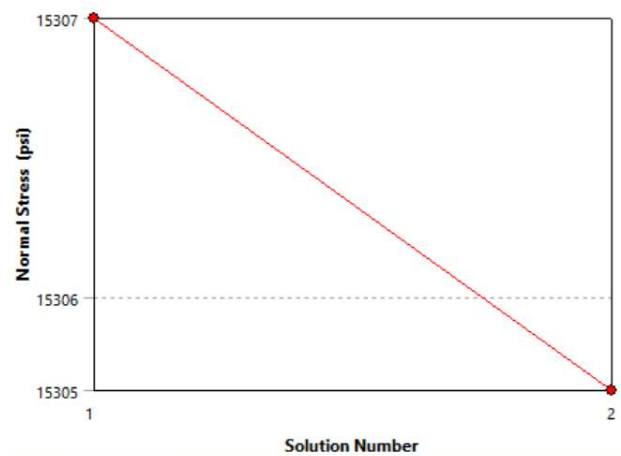

Figure 8: Convergence Plot, 0.75 in

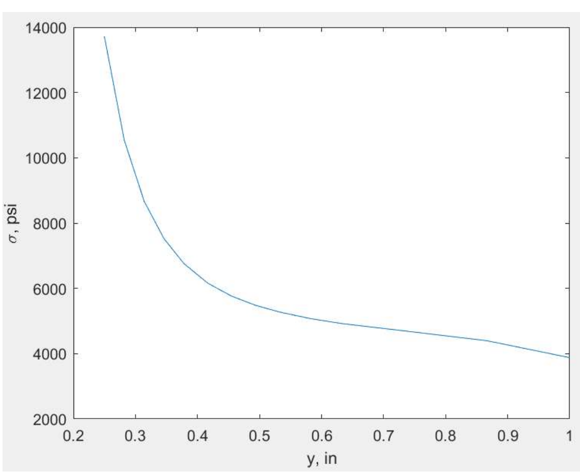

Figure 9: Stress Distribution, 0.5 in

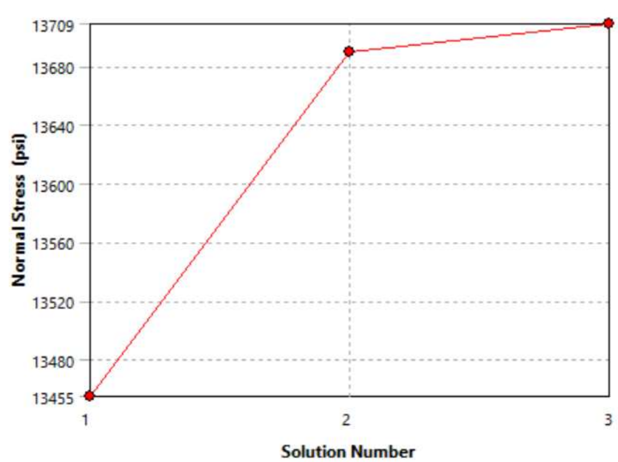

Figure 10: Convergence Plot, 0.75 in

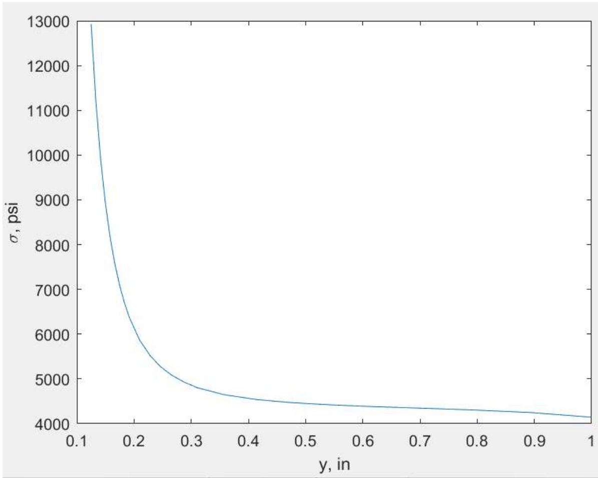

Figure 11: Stress Distribution, 0.25 in

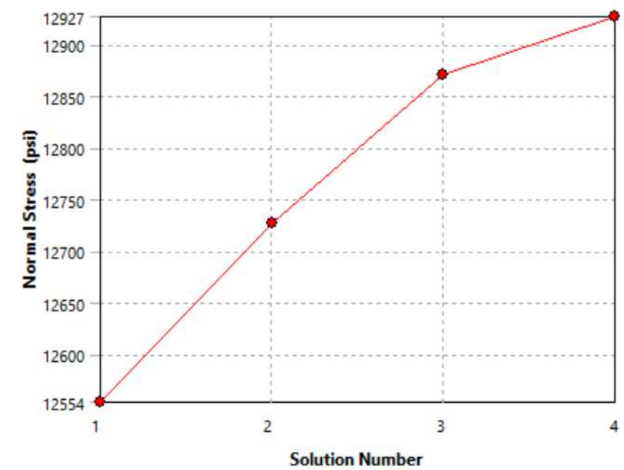

Figure 12: Convergence Plot, 0.25 in

It appears that the hole diameter does not have an effect on the number of iterations required for ANSYS to converge on a solution. Looking at the stress distributions, the stress has a steeper drop off with distance away from the hole for smaller hole diameters. This means that the hole causes a stress concentration on a smaller area of the plate, which makes sense. This is an example of Saint Venant's Principal. For the 1 inch diameter hole, there is no point sufficiently far away from the stress concentration where the nominal stress value is reached. For the 0.25 inch diameter hole, a majority of the width of the plate is sufficiently far away and a constant nominal stress value is reached.

To determine the validity of the ANSYS results, the maximum normal stress calculated by ANSYS was compared to a theoretically calculated value. The theoretical value was obtained by calculated the stress concentration factor using the hole diameter and width of the plate. The stress concentration factor, along with the nominal stress, can then be used to find the theoretical maximum stress. This calculation was performed as follows:

$$K_t = 3 - 3.14\left(\frac{d}{D}\right) + 3.667\left(\frac{d}{D}\right)^2 - 1.527\left(\frac{d}{D}\right)^3$$

$$\sigma_{nom} = \frac{P}{t(D-d)}$$

$$\sigma_{max} = \sigma_{nom} K_t$$

The results are listed below.

Hole Diameter, in	1	0.75	0.5	0.25
Theoretical K_t	2.1559	2.2576	2.4203	2.6618
ANSYS K_t	2.1607	2.2575	2.4265	2.6694
Theretical σ_{max}, psi	18270.127	15306.078	13674.17	12890.1426
ANSYS σ_{max}, psi	18311	15305	13709	12927
% Difference in σ_{max}	0.224	0.007	0.255	0.286

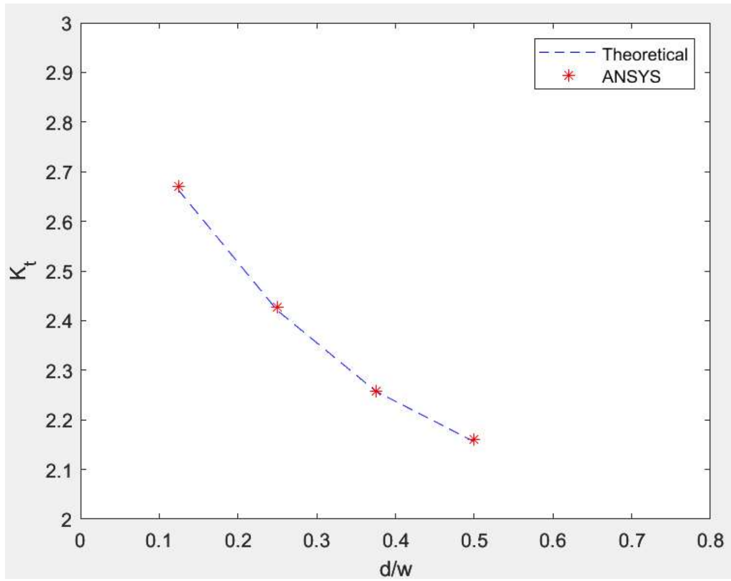

Figure 13: Stress Concentration Factors

The results are very similar to that predicted by theory, resulting in very low percent differences. ANSYS seems to capture the effects of stress concentration very well. In this analysis, a convergence object was added to the normal stress solution. This allowed for a very refined element size near the hole and larger size far from the hole. Therefore, very accurate results are produced as this mesh size is needed near the stress concentration. Using both the convergence and symmetry tools allowed for very accurate results that could not be obtained otherwize.

The plot of the ANSYS K_t values matches the theoretical very well. The value of K_t varies inversely to the maximum stress. As the hole diameter increases, K_t decreases. This makes sense as the nominal stress will be significantly larger when there is a larger hole as the cross sectional area is much smaller. It seems that the theoretical K_t values can be a very useful tool in predicting the maximum stress.

Conclusions:

The ANSYS results confirm that the normal stress reaches its maximum near the hole and decreases to an average stress value at a sufficient distance away from the stress concentration. As the diameter of the hole increases, the maximum stress value also increases and the effect of stress concentration spreads further along the plate. Consequently, the plot for the 1 inch hole does not level out completely, whereas the effect of stress concentration diminishes quickly for smaller holes. This is consistent with Saint Venant's Principle, which states that the stress depends on the static resultant rather than the stress distribution when sufficiently far from the stress concentration. The maximum stress value for each plate was over twice as large as the nominal stress, indicating the substantial impact of stress concentration that cannot be accurately predicted by a general stress formula.

To ensure high efficiency and accuracy, this analysis utilized ANSYS's symmetry tool and convergence object. The symmetry tool allowed for analyzing only one quarter of the element, while still obtaining the same results as analyzing the entire element. The convergence object ensured that a solution was found by refining the mesh near the stress concentration and using a larger mesh away from it. This approach enabled ANSYS to model the structure with high precision. When compared to theoretical results, the error produced by ANSYS was found to be very small, demonstrating its potential as an extremely useful tool for modeling the effects of stress concentration. It is critical to understand the concept of stress concentration to predict structure failure accurately. By utilizing ANSYS, it is possible to identify potential stress concentration areas and make necessary design modifications to ensure structural integrity.

References:

[1] Hibbeler, R. C. (2016). *Mechanics of Materials*. Pearson.
[2] Megson, T. H. G. (2013). *Aircraft Structures for engineering students*. Butterworth-Heinemann.

Appendix:

```
clc
clear
close all
kt = [2.1559
2.2576
2.4203
2.6618];
kta = [2.1607
2.2575
2.4265
2.6694];
d = [1
0.75
0.5
0.25];
figure(1)
plot(d./2,kt,'b--')
hold on
plot(d./2,kta,'r*')
xlabel('d/w')
ylabel('K_{t}')
ylim([2 3])
xlim([0 0.8])
legend('Theoretical','ANSYS')
hold off
D25 = importdata('0.25data.txt');
y25 = D25.data(:,4);
sig25 = D25.data(:,7);
sig25 = sort(sig25,'descend');
y25 = sort(y25,'ascend');
figure(2)
title('Normal Stress Distribution, D = 0.25 in')
plot(y25,sig25)
xlabel('y, in')
ylabel('\sigma, psi')
D50 = importdata('0.5data.txt');
y50 = D50.data(:,4);
sig50 = D50.data(:,7);
sig50 = sort(sig50,'descend');
y50 = sort(y50,'ascend');
figure(3)
```

```
title('Normal Stress Distribution, D = 0.5 in')
plot(y50,sig50)
xlabel('y, in')
ylabel('\sigma, psi')
D75 = importdata('0.75data.txt');
y75 = D75.data(:,4);
sig75 = D75.data(:,7);
sig75 = sort(sig75,'descend');
y75 = sort(y75,'ascend');
figure(4)
title('Normal Stress Distribution, D = 0.75 in')
plot(y75,sig75)
xlabel('y, in')
ylabel('\sigma, psi')
D1 = importdata('1data.txt');
y1 = D1.data(:,4);
sig1 = D1.data(:,7);
sig1 = sort(sig1,'descend');
y1 = sort(y1,'ascend');
figure(5)
title('Normal Stress Distribution, D = 1 in')
plot(y1,sig1)
xlabel('y, in')
ylabel('\sigma, psi')
```