Homework 5

Anshul Yadav

November 5 2021

Abstract

In this report I analyse a plate with a square hole that is subjected to 2 pressure loads. A biaxial pressure of 250 KPa is applied on its 2 sides and a pressure of 500 Pa applied on top of it. An Ansys model was made using shell elements (Tet 281).

Method:

The problem

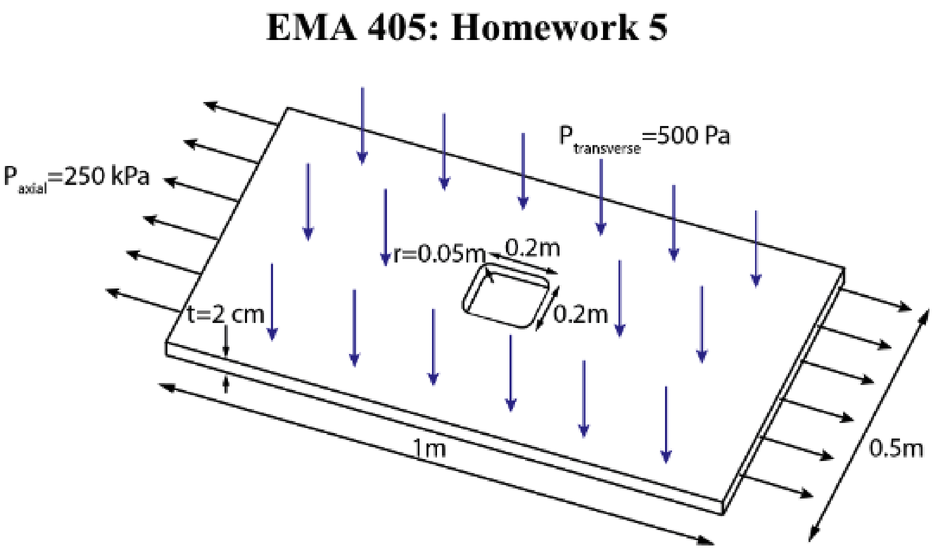


Figure 1: The problem

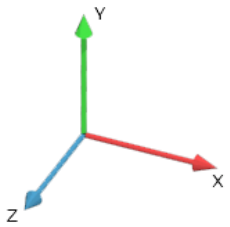


Figure 2: The coordinate axis

The problem can be thought of as a superposition of the axial load of 250KPa and the transverse load of 500 Pa. This is indeed true since the plate is only restricted motion in the y direction. Thus the axial load produces a normal stress and the transverse load produces a bending.

Axial Load:

The max stress due to the axial load is found using stress concentration equations that were presented in the problem.

Factor of stress concentration k for various dimensions

$$\sigma_{max} = k\sigma_1 \quad \text{and} \quad k = K_1 + K_2\left(\frac{b}{a}\right) + K_3\left(\frac{b}{a}\right)^2 + K_4\left(\frac{b}{a}\right)^3$$

where for $0.2 \le r/b \le 1.0$ and $0.3 \le b/a \le 1.0$

$K_1 = 14.815 - 15.774\sqrt{r/b} + 8.149r/b$
$K_2 = -11.201 - 9.750\sqrt{r/b} + 9.600r/b$
$K_3 = 0.202 + 38.622\sqrt{r/b} - 27.374r/b$
$K_4 = 3.232 - 23.002\sqrt{r/b} + 15.482r/b$

Figure 3: Stress Concentration Equations.

$K_1 = 14.815 - 15.774\sqrt{0.05/0.1} + 8.419 * 0.05/0.1 = 7.871$ (eq1)

$K_2 =- 11.201 - 9.75\sqrt{0.05/0.1} + 9.600 * 0.05/0.1 =- 13.295$ (eq2)

$K_3 = 0.202 + 38.622\sqrt{0.05/0.1} - 27.374 * 0.05/0.1 = 13.825$ (eq3)

$K_4 = 3.232 - 23.002\sqrt{0.05/0.1} + 15.482 * 0.05/0.1 =- 5.292$ (eq4)

$K = K_1 + K_2 b/a + K_3 b^2/a^2 + K_4 b^3/a^3 = 7.871 - 13.295 + 13.825 - 5.292 = 3.109$ (eq5)

$\sigma_{max} = K * 250KPa = - 3.109 * 250KPa = 777.25KPa = .77725\,MPa$ (eq6)

σ_{max} is the max in plane normal stress.

Normal Load:

For the normally applied pressure the problem asks us to ignore the effect of the hole and to change the problem into a normally distributed vertical load one. Thus, we have a rectangular plate that is simply supported on both ends subject to a normal load.

$\omega = 500 * 0.5 = 250\,N/m$ (eq7)

Then the variation of bending moment with distance from the left end is given by (L = 1m):

$M(x) = - \omega * (x^2/2 - Lx/2) = - 250 * (x^2/2 - x/2)$ (eq8)

Eq8 was obtained from EMA 303. The max value of M(x) is 31.25 Nm at x= 0.5.
The value of max tensile stress is :

$\sigma = My/I = 31.25 * 0.01/(1/12 * 0.02^3 * 0.5) = 937.500KPa$ (eq9)

The max compressive stress is -$937.500 KPa$.

Adding the two solutions we get max Tensile stress as 1.714 MPa.

The max compressive stress occurs far away from the cutout. This is because far away from the cutout the effect of the cutout is not felt so the max compressive stress which is the difference in stress due to axial and bending stresses is minimised. At the absolute furthest distance we have,

$\sigma_{compressive,max} = \sigma_{axial,min} - (\sigma_{bending,max}) = 250KPa - 937.5KPa = -687.5KPa$

Thus the magnitude of max compressive stress is 687.5 KPa

This occurs far from the cutoff at the midway point.

ANSYS Solution:

A Finite element model was made to emulate ¼ of the total specimen.

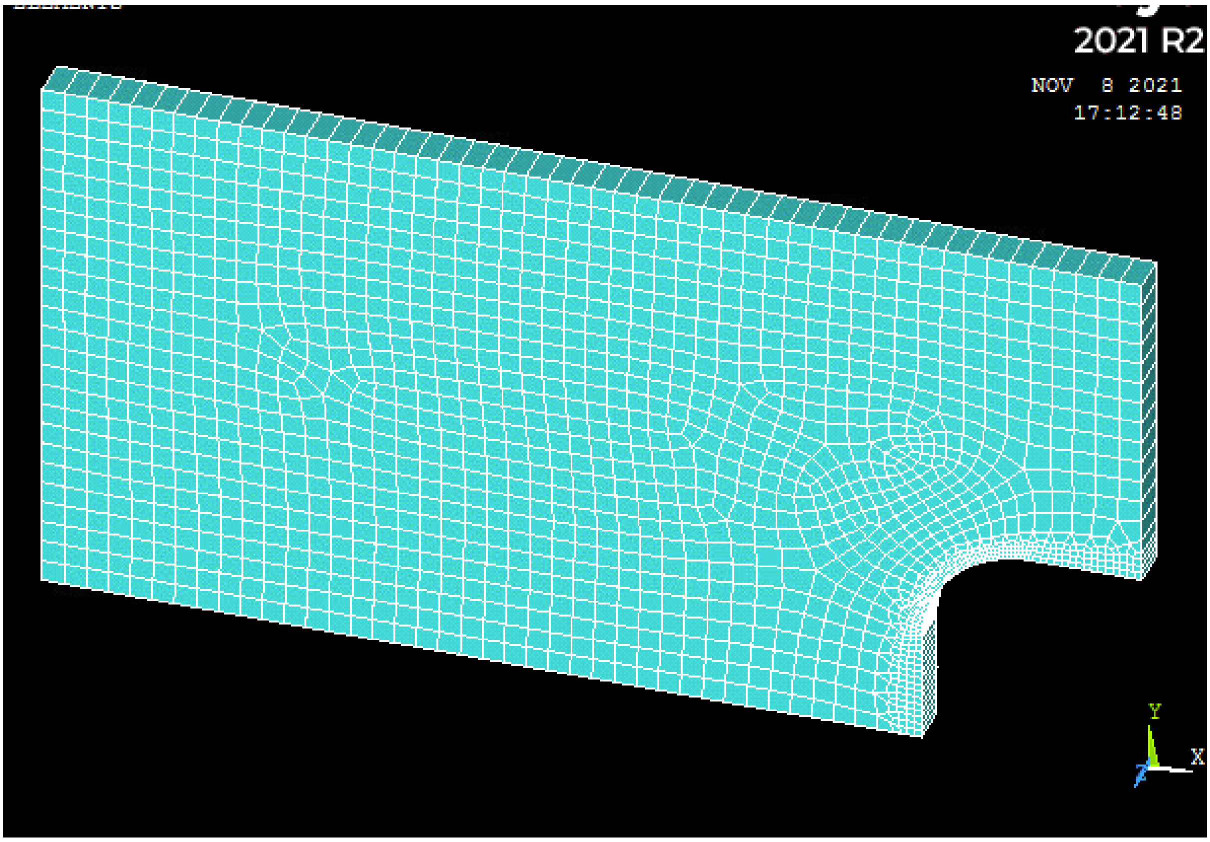


Figure 3: FEA model with mesh

A finer mesh density was used near the cutout to account for stress concentration.

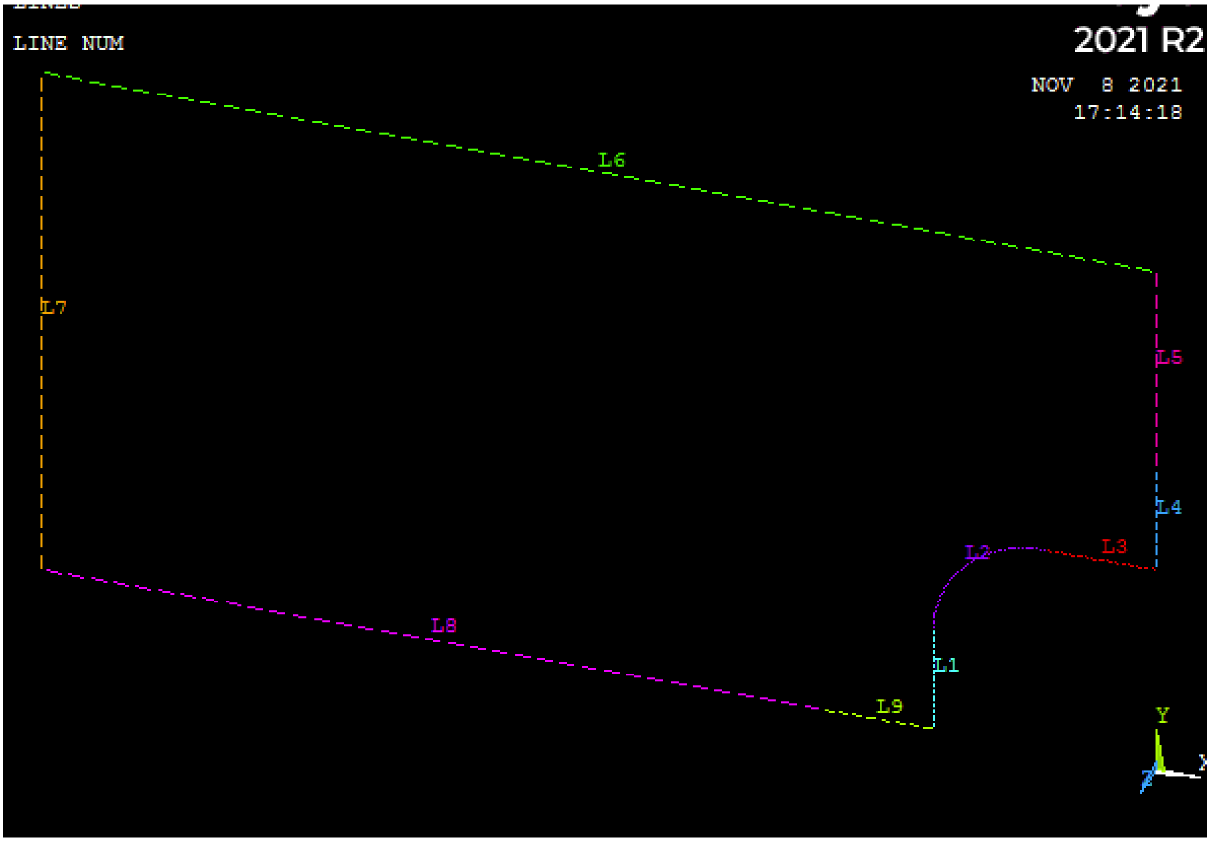


Figure 4: Model from Lines.

Symmetric boundary conditions were applied on lines 8,9,4,5. Displacement was restricted on line 7 in the y direction. An areal pressure was applied on the area in the -z direction (500 Pa). A line load was applied on line 7 of (-5000 N/m) using the eqn 10.

$$Line\ Load\ =\ 250KPa\ *\ 0.02m\ =\ 5000N/m$$

(eqn10)

Discussion of Results.

The following Stress values were measured on the 3 surfaces (top, middle, bottom).

Plots of Stress variation in x direction:

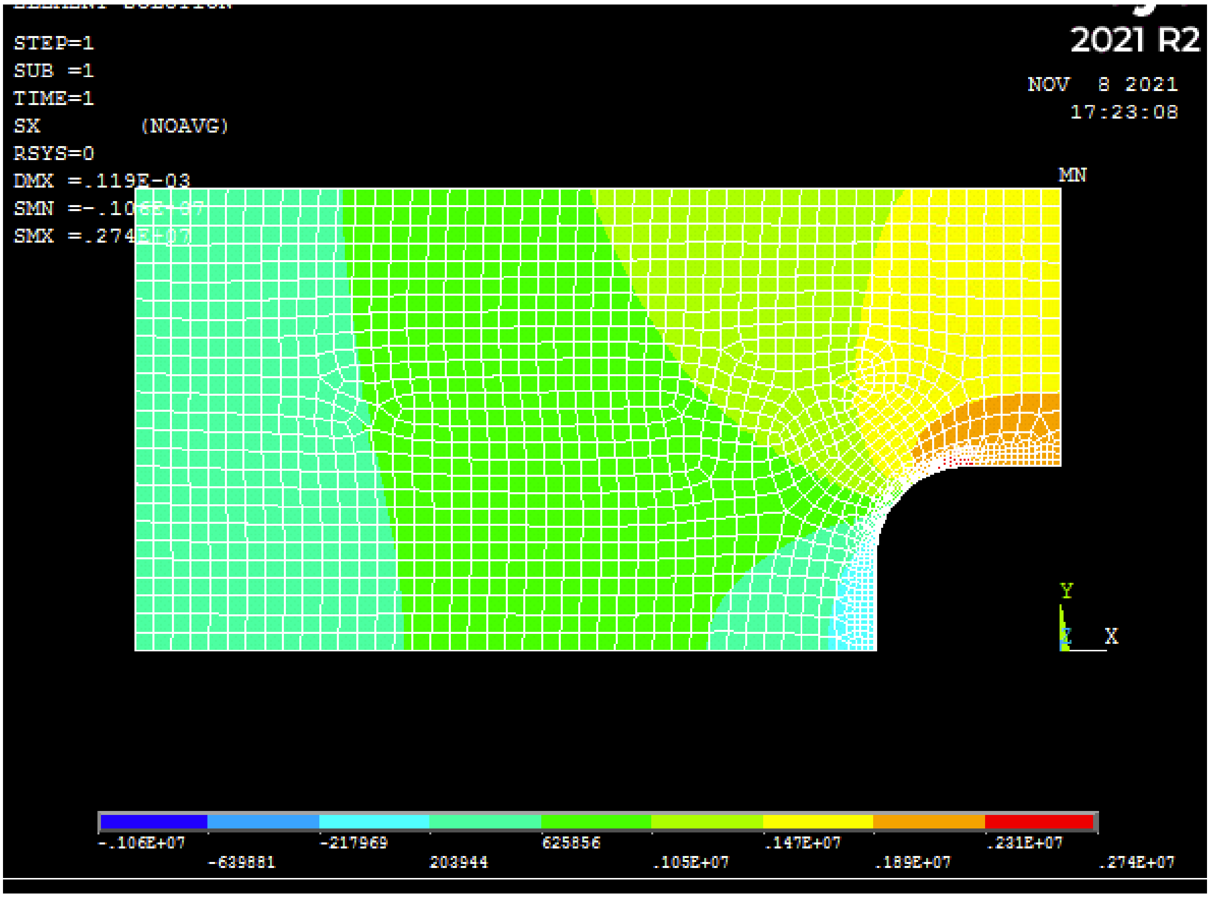


Figure 5: Variation of Stress

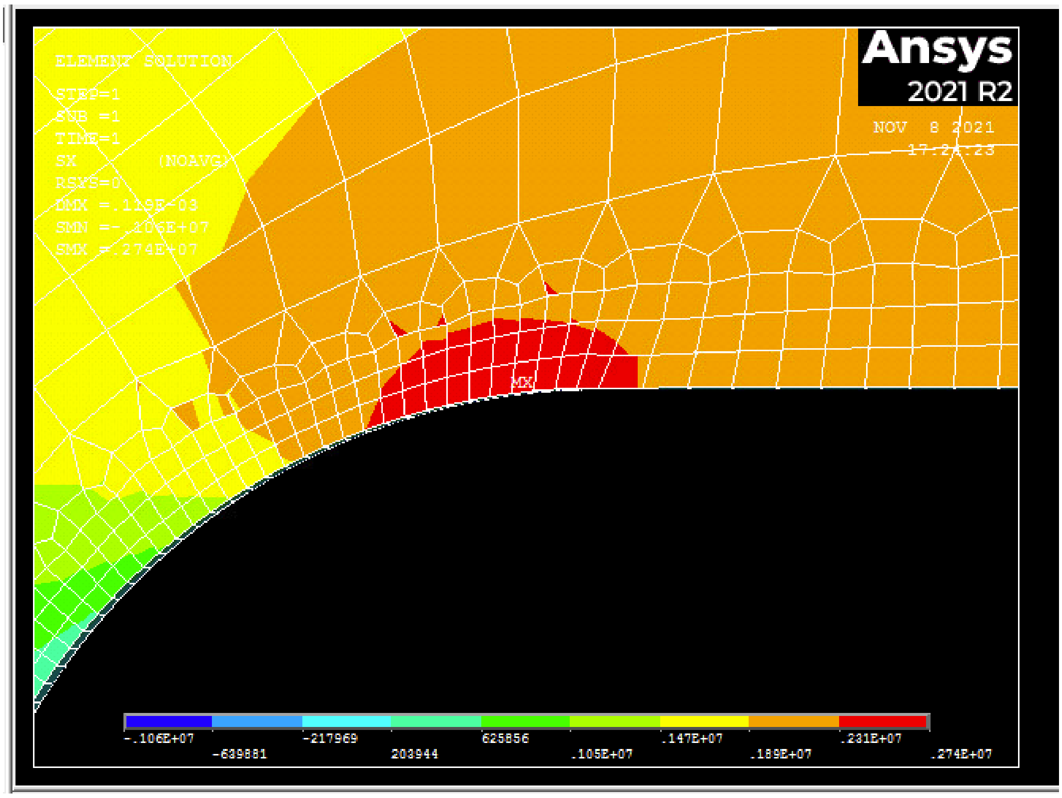


Figure 6: Variation of Stress close to the hole

Results at the Bottom Layer

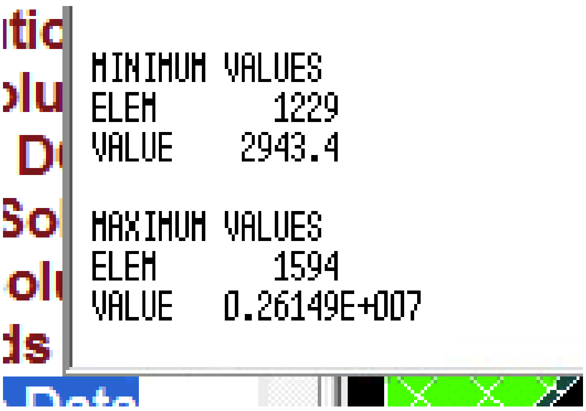


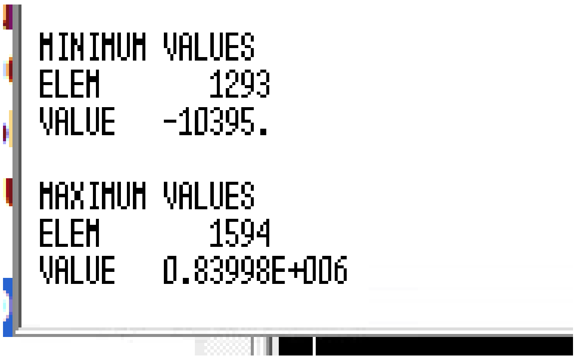
Figure 7: Results for the Bottom Layer

The max tensile stress on the bottom layer was found to be 2.61MPa, which was near the cutout as seen in figure 6. The theoretical maximum is 1.71 MPa.
There are several reasons that account for this difference.

- First and foremost the stress concentration (eqn6) is for an infinite plate with a rectangular hole. The plate used in modelling is not infinite.
- Secondly, ANSYS is an approximation solver so its values should be taken with a grain of salt. This is especially true since the stress gradient near the cutout is large.
- Lastly, the theoretical calculation for the bending stresses excluded the effect of the cutout, which may explain why the difference is so much

Since we don’t expect compression to occur at the bottom layer, the minimum value is not of any physical significance.

Results at the Middle Layer:

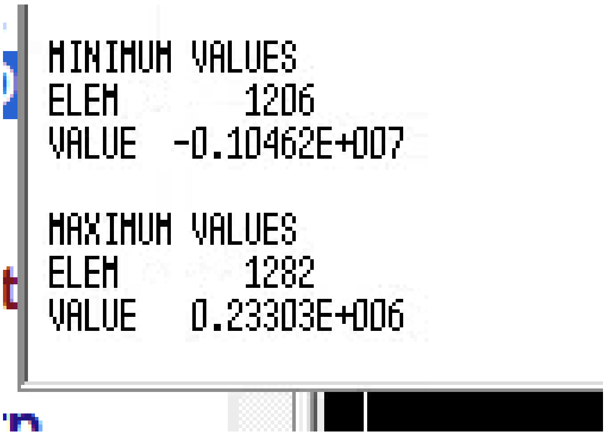

MINIMUM VALUES
ELEM 1293
VALUE -10395.

MAXIMUM VALUES
ELEM 1594
VALUE 0.83998E+006

Figure 8: Results for the middle layer

The max tensile and the max compressive are 840KPa and 11.4 KPa. Since we don't expect the theoretical max/min values to occur at the mid plane, these values are not of much physical significance.

The Bottom Layer


MINIMUM VALUES
ELEM 1206
VALUE -0.10462E+007

MAXIMUM VALUES
ELEM 1282
VALUE 0.23303E+006

Figure 9: Results of the Top Layer

A max value of 1.046 MPa was observed under compression (global minimum) on the top layer, far away from the cutout. The location of the point that is predicted by theory is vindicated by ANSYS, even though the magnitudes differ. Theoretical max compressive stress value is 687.5KPa.

There are several reasons for this difference.

- First and foremost the stress concentration (eqn6) is for an infinite plate with a rectangular hole. The plate used in modelling is not infinite.
- The theoretical calculation for the bending stresses excluded the effect of the cutout, which may explain why the difference.
- A coarse mesh density was used to simulate stresses far from cutout. A finer mesh density may yield better results.

Conclusion:

In conclusion the stress gradient near the cutout is not accurately modelled. Even though the signs of the stresses were the same as the theoretical predictions, the magnitudes were different. The difference in magnitude between theory and ANSYS never differed by more than a factor of 2.

References:

http://www.matweb.com/search/datasheet.aspx?matguid=7f38db56864e46659a38760e6de4a5db&ckck=1

https://www.azom.com/properties.aspx?ArticleID=969