Sorbonne Université

PW2 - Perforated plate

Authors :
Alexandre RICHARD
Valentin Duvivier

Supervisors :
Sylvain Patinet
Rémi Cornaggia

December 28, 2020

Contents

1 Introduction

We consider a structure purposely perforated. Therefore, we aim to determine the stresses suffered by such a structure. The purpose is then to, when building it physically, dimensionalize the plate given the loads apply to it.

Such a plate can be found under below forms :

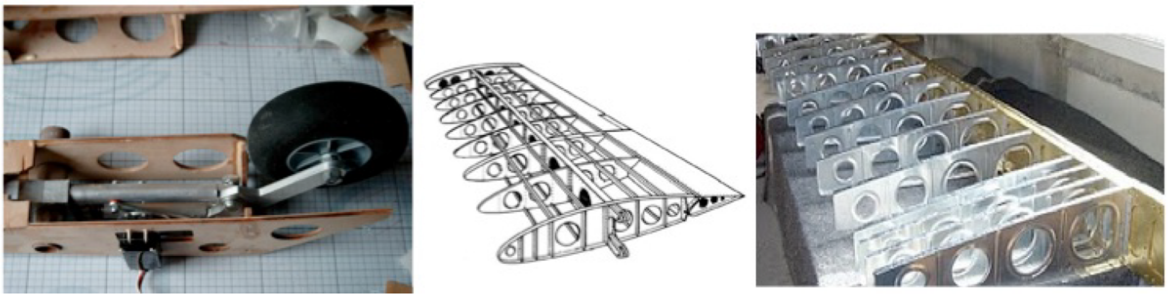


Figure 1: Physical use - perforated plate

For simplification sake, we assume a symmetric load.

Domain

We assume a square plate whose dimensions are big with respect to its hole radius. This assumption ensures no interaction and dependence between hole and sides.

2 Problem definition

To respect above consideration on the structure's dimension, we consider the following data :

- $L = 40$ cm
- $a = 2.5$ cm

One can see that $\frac{a}{L} \approx 0.125 \ll 1$. We assume these length as conform and a visual analysis of stress fields will simply confirm this choice.

For what is up to the structure's composition, we assume it to be made of an isotropic, homogeneous, linear elastic material whose mechanical properties are as follows :

- $E = 135$ GPa
- $\nu = 0.35$

Moreover, as mentioned during introduction, the structure is suffering a symmetrical load. We then impose a force $f_s = 1$ MPa applied as follows :

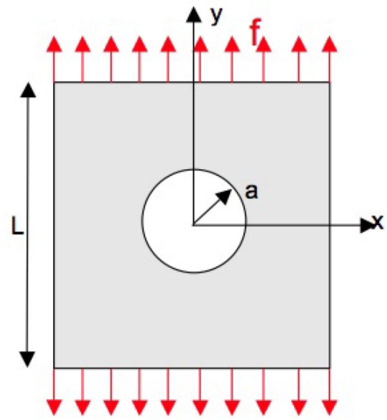


Figure 2: Structure in Cartesian plan

3 Preliminary solution

This preliminary part is an introduction to the problem on a theoretical aspect.

1. Uniqueness :

   As that, one can note the problem is ill-defined. Indeed, the displacement one could get from an analysis would be given within a rigid body, namely there is an infinite number of solution. What we are to do then is to impose some displacement on the mesh, using more particularly its symmetries.

   One can see that from a geometrical analysis of the structure itself and the load applied to it, the structure will present symmetries. Here, we have in fact 2 axes of symmetries : $\Sigma_{x=0}$ and $\Sigma_{y=0}$.

   Such a simplification implies one can reduce the mesh to a quarter of the current one. We indeed reduce it to a half when considering $\Sigma_{x=0}$ symmetry and a half of an half when considering $\Sigma_{y=0}$ symmetry.

   In the end, the mesh one will study has following form :

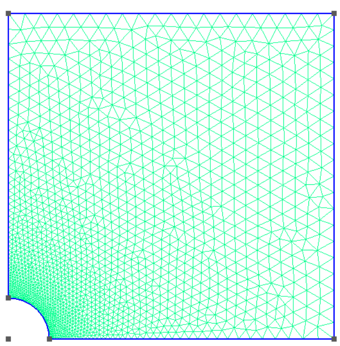


Figure 3: Simplified mesh

To finish, this type of symmetry implies that faces in $\Sigma_{x=0}$ and $\Sigma_{y=0}$ mustn't move in their normal direction. Therefore, this is equivalent to say that we must impose displacements on two sides and that the problem is now well-defined, i.e it brings unique solution in displacement. For what is up to the stress field, the load applied on a third side and the zero load applied on the last side brings a unique solution in stress.

2. Mesh's considerations

   First and foremost, we recall the geometrical consideration made in *introduction* and *problem definition* parts that is the hypothesis of infinitely big length of the square with respect to the hole's radius.

   A condition of plane elasticity derives as well from this dimensional consideration.

   Finally, the type of element is here imposed : we work on T6 elements. This consideration is natural when provided that for our analysis the stresses suffered by the structure are inhomogeneous, namely sudden variation in the stress field for element alongside.

3. Stress fields

   We know that the crack in such a case occurs in the planes orthogonal to principal normal stresses if assuming mode 1 failure. Thereby, in our case, cracks will appear along the hole. Indeed, stresses at boundaries are applied along $\underline{e_y}$ axis, and so cracks will develop due to elongation and compression, occurring respectively on $\underline{e_y}$ and $\underline{e_x}$ axes.

   In the end, crack will occur preferentially along the circle in $\underline{e_x}$ direction while stresses might grow along increasing y as we then get closer to applied forces.

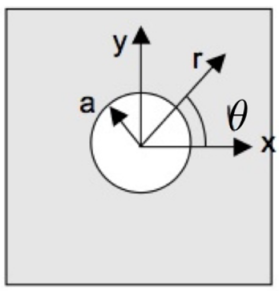
   To go further on this point, we are to do an analysis of stress fields.

4 Stress field accuracy

From mesh's consideration, we are able to get a simple enough problem to derive analytical results. Therefore, a point of this section will be to compare numerical and analytical results regarding stress field. Here are the equations and associated framework :

$$\sigma_{rr} = \frac{f}{2}\left(1 - \frac{a^2}{r^2}\right) + \frac{f}{2}\left(1 - \frac{4a^2}{r^2} + \frac{3a^4}{r^4}\right)\cos\left(2\left[\theta - \frac{\pi}{2}\right]\right)$$
$$\sigma_{\theta\theta} = \frac{f}{2}\left(1 + \frac{a^2}{r^2}\right) - \frac{f}{2}\left(1 + \frac{3a^4}{r^4}\right)\cos\left(2\left[\theta - \frac{\pi}{2}\right]\right)$$
$$\sigma_{r\theta} = -\frac{f}{2}\left(1 + \frac{2a^2}{r^2} - \frac{3a^4}{r^4}\right)\sin\left(2\left[\theta - \frac{\pi}{2}\right]\right)$$

(a) Analytical equation

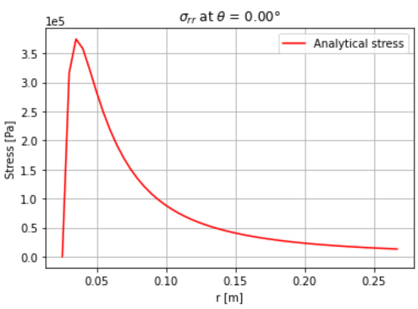


(b) Structure in Polar plan

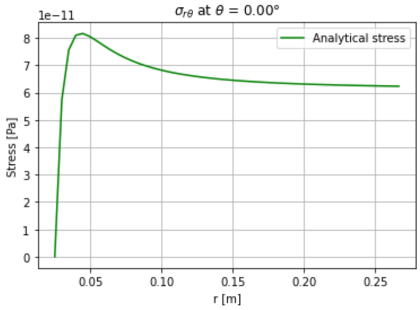
4. Computation analytical results

As mentioned above, we get analytical results that one print for further analysis :

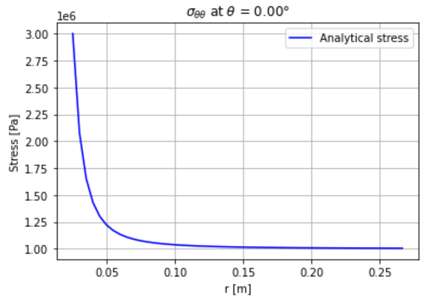
$\theta = 0$:



(a) Analytical equation

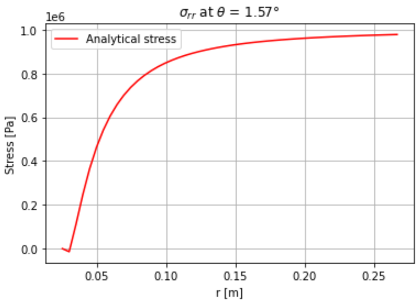


(b) Structure in Polar plan

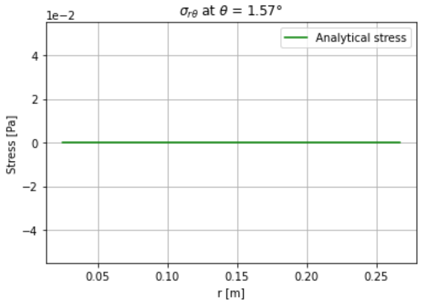


(c) Structure in Polar plan

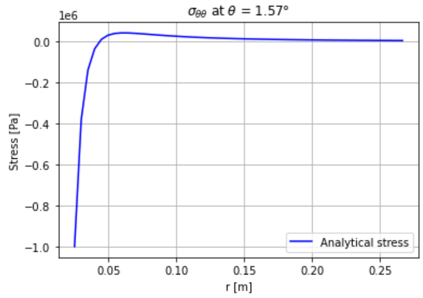
$\theta = \frac{\pi}{2}$:



(a) Analytical equation



(b) Structure in Polar plan



(c) Structure in Polar plan

Note :

- At $\theta = 0$: $\sigma_{rr} \equiv \sigma_{xx}$ and $\sigma_{\theta\theta} \equiv \sigma_{yy}$
  - σ_{rr} and $\theta = 0$: the stress is maximum along $\underline{e_r}$ at the circle (low radius). This confirms the observation on cracks (intensity and direction).
  - $\sigma_{r\theta}$ and $\theta = 0$: given formula of $\sigma_{r\theta}$, at low angles, sinus behaves as θ and so we have a non-null, but very low, stress.
  - $\sigma_{\theta\theta}$ and $\theta = 0$: once again, the stress is maximal at the circle edge but it gets lower for increasing r. This is what one would expect when far from load line application.

To summarize : the cracks occur along normal axes $\underline{e_x}$ and close to the circle. These stress result from applied load and show with ease where max stresses are.

On the other hand :

- At $\theta = \frac{\pi}{2}$: $\sigma_{rr} \equiv \sigma_{yy}$ and $\sigma_{\theta\theta} \equiv \sigma_{xx}$
  - σ_{rr} and $\theta = \frac{\pi}{2}$: stresses are mainly influenced by above load along same axis (i.e. $\underline{e_y}$). As σ_{rr} is the stress developed along $\underline{e_y}$, it's not the axis where crack take place as expected.
  - $\sigma_{r\theta}$ and $\theta = \frac{\pi}{2}$: stresses are null as expected from formula.
  - $\sigma_{\theta\theta}$ and $\theta = \frac{\pi}{2}$: similarly stresses are mainly influenced by load on upper surface, even if, as we are considering stresses along crack axis, we can see that for low radius a pick of stress appears. This finally confirms max stress occurring at circle in $\underline{e_x}$ direction.

It has here been implied that cracks take place where max stress is.

5. Stress at nodes

   We now want to get numerical stress values in an aim of comparison.

   A first point is to extract stresses values from a code solving numerically the problem. Therefore, given the structure's data and the load applied to it, we obtain several results such as displacement, stress, etc. However, as we work on T6 elements, some of the field such as stress fields are evaluated at nodes.

   Thereby, one might want to extrapolate values from Gauss points to the nodes. This part being provided, we've been building a function called *extractSig(self, mesh, V, theta, ...)* that return stress field at nodes for a given theta, provided the stress field at Gauss point.

   A part of the so-called code is provided here below :

```python
def extract_Sig(self, mesh, V, theta):

    """ Sigma extraction """

    for (j,e) in enumerate(mesh.elem_list):
        for i in range(ngauss)])
            V_node = np.linalg.lstsq(N,V[ngauss*j:ngauss*(j+1)])[0]
            for ind_co in range (node_coords.shape[0]):
                co = node_coords[ind_co]

                if np.abs(np.arctan2(co[1],co[0])-theta ) < 0.02:
                    co_rad=np.sqrt(co[0]**2+co[1]**2)

                    nodes_rad_coord.append(co_rad)    # RADIUS
                    V_extract.append(V_node[ind_co]) # STRESS FIELD
```

Listing 1: ExtractSigma function

The main points of this code is that it brings a new feature of extrapolation to nodes (through *np.linalg.lstsq*) and that it will allow us to get stress components at asked angles theta (through $V_{extract}$).

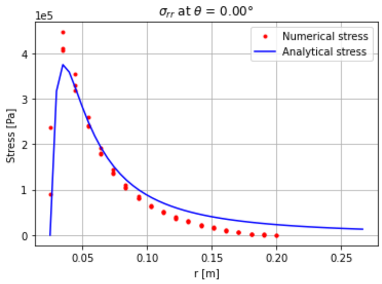
To finish, a point we had to be careful about has been to consider the good set of nodes. We indeed work on T6 elements but we nevertheless only know stresses at triangle's vertices. Therefore, we had to consider only three nodes and reshape consequently our vectors.

The results developed in this sub-section are visually presented in next part.

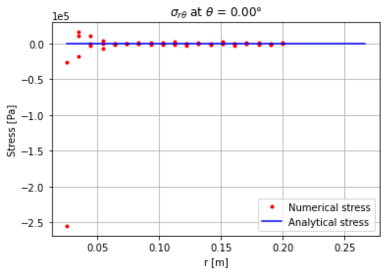
6. Comparison Analytical/Numerical results

   At the aim of comparing our graphs, we build a second method called *plotExtractSig*, displaying superposed curves at a given angle.

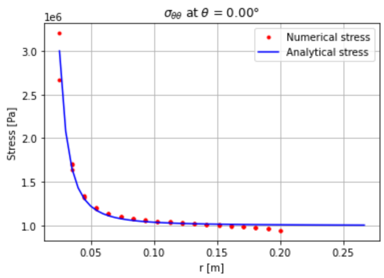
   Here are the resulting graphs :



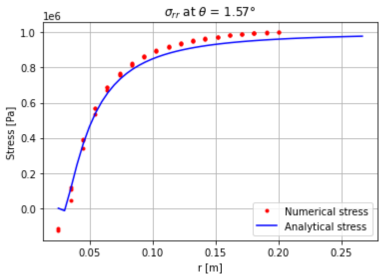
(a) Analytical equation



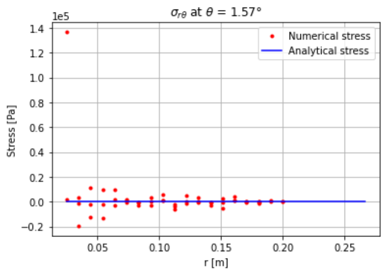
(b) Structure in Polar plan



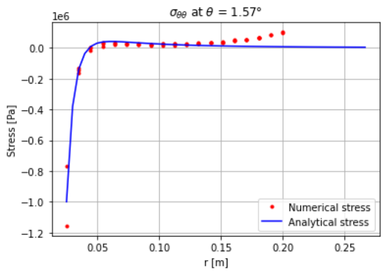
(c) Structure in Polar plan



(a) Analytical equation



(b) Structure in Polar plan



(c) Structure in Polar plan

One can see numerical results are given within a range of r. This implies that we can get stresses at middle range but at edges liability is lower. However, the stress fields are well approximated.

5 Ellipsoidal holes

7. Concentration factor

   We define the stress concentration factor K_G as :

$$K_G = \frac{\sigma_M}{\sigma_G}$$

with σ_M the max stress at hole and σ_G stress far from hole. For this last variable, we assume the nominal stress to be defined as :

$$\sigma_G = \frac{F}{A} = f_s$$

Using the code provided below, we then get the sought factor K_G :

```
# K_G

N     = 50
r     = 0.025
theta = np.linspace(0, np.pi/2, N)

Stresses = np.zeros((3, N))
Siggg = [list_res.Sigxx, list_res.Sigxy, list_res.Sigyy]

for i in range(len(Siggg)):
    for j in range(len(theta)):
        (R, V_num) = list_res.extract_Sig(mesh, Siggg[i], theta[j])
        Stresses[i,j] = V_num[R==r]

sigma_G = fs/(Length/2)
sigma_M = np.max(Stresses)
K_G     = sigma_M/sigma_G
```

Listing 2: ExtractSigma function

The purpose here is to test every angle theta to then get the max stress in $r = a$, giving therefore σ_M.

We finally get, for a circle :

$$K_g \approx 3.06$$

To have more hindsight on this value, we will compare it to the ones of ellipse hole cases.

8. Elliptic hole

   One consider an ellipse instead of a circle whose ratio $\frac{a}{b}$ equals 2.

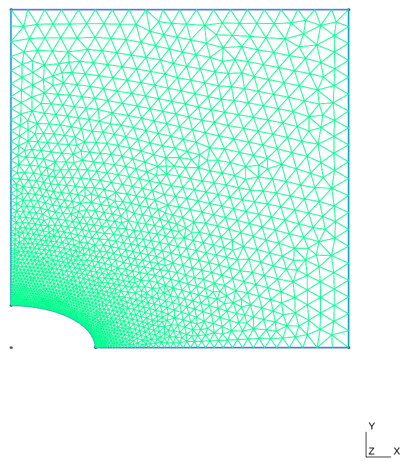


Figure 4: Ellipsoidal mesh - ratio 2

From a similar process, we get :

$$K_g \approx 5.42$$

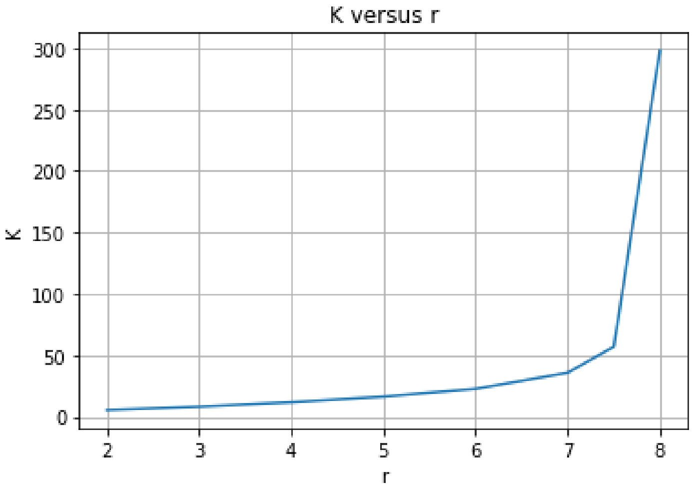
We get a ratio whose value is higher, meaning the max stress with respect to nominal stress is higher in ellipsoidal case. As we've been considering same forces, we have the same σ_G and so it implies the maximum stress is bigger in the ellipsoidal case.

We can extrapolate on the effect of either the bending of the hole or its size. While a bigger hole would imply a non perfect case (no influence from the edges), the case of a flat hole would imply a crack shape. We will develop this last point by studying ellipse holes for different $\frac{a}{b}$ ratios.

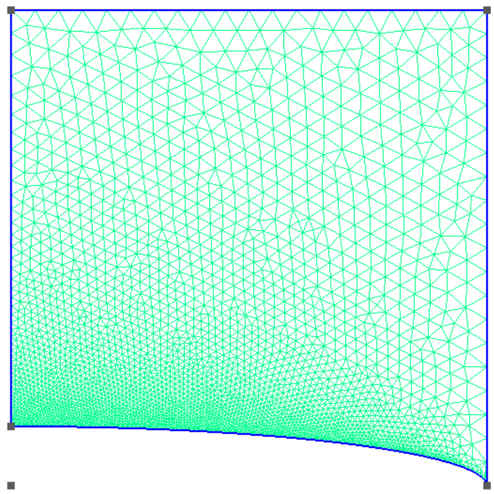
9. Limit case

   As we've highlighted the bending (though $\frac{a}{b}$ ratio) had influence, and we want to confirm this point by studying limit case where the ellipse flattens.

Here below is a graph made of different ratios $\frac{a}{b}$:



(a) Ellipsoidal mesh - several ratios



(b) Ellipsoidal mesh - ratio 8

The left hand side graph implies the maximum stress inside the mesh increases with the ellipse flattening. This behavior brings out that failure are bigger and take place much quicker when the distance hole-wall decreases.

In such a case, the width decreasing a lot and the hypothesis of no interaction hole-edge being gradually unverified, we get a limit case where one get an infinitely increasing stress with r increasing.

The geometry such as the one presented in figure (b) thus presents high stresses at the ellipse right hand side and potentially high failure as well.

6 Interacting holes

So far, we've reduced the geometry of the initial plate using its symmetries. We will do this again but in order to study interactions inside the mesh.

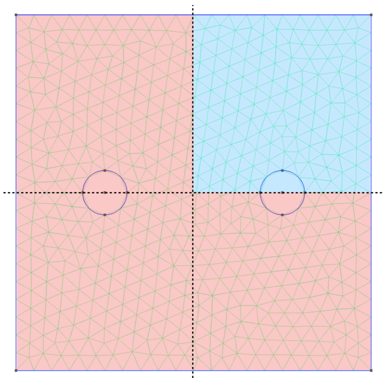
10. 2 holes

We aim to study the interaction in a plate of several holes. For simplification sake, we are studying the interactions between 2 holes.

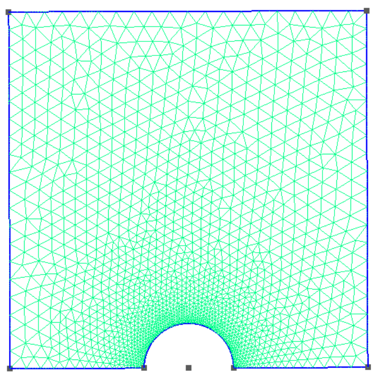
We know we won't have to model every holes, but that we will rather use symmetry planes. Then, to go from a centered hole to 2 interacting hole, one needs to consider a new plane of symmetry. What we have to be careful about is the boundary conditions on displacement that we imposed to ensure uniqueness.

We have indeed seen we can have symmetry planes but that implies we must impose BC on displacement here. Then, as we won't impose any further BC, we will have to deal with the existing one.

We present here a graph recalling previous symmetry and presenting the new one :



(a) Mesh of a plate with 2 holes



(b) Top right hand side square

We have 4 bigger squares, whose top-right one (blue one) is the one we've been studying so far. Moreover, this is the one we consider again, even though we now offset the hole to the middle center.

Let's now go through a few graphs showing interactions occur. Firstly, we show the case $\theta = 0$ for σ_{yy}.

We display here a graph showing that the analytical equations are not verified anymore :

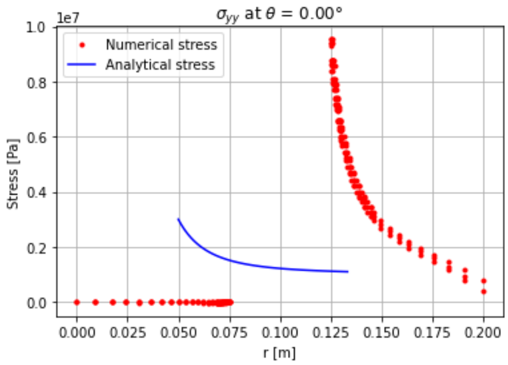
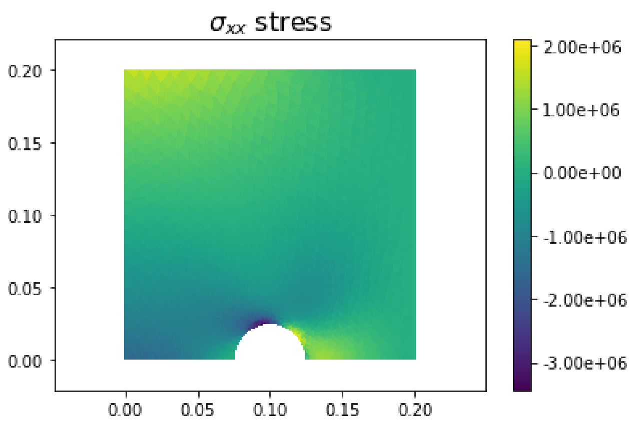
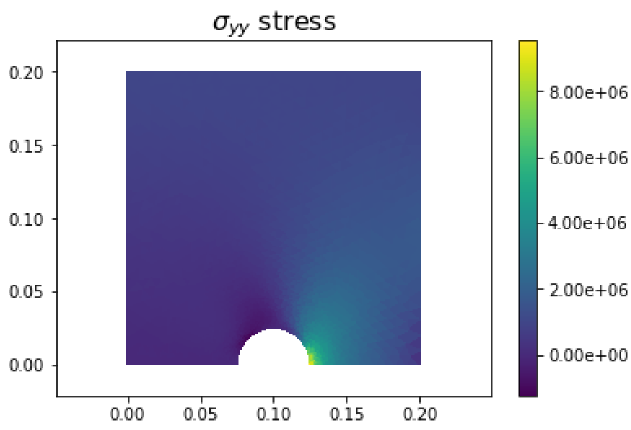


Figure 5: Mesh of a plate with 4 holes

Furthermore, what changes with hole interaction is that the stress fields will differ. Indeed, to consider more hole will imply stress from one hole to an other, which might result in higher failure. Here below is a graph of normal stresses along axes $\underline{e_x}$ and $\underline{e_y}$:



(a) σ_{xx}



(b) σ_{yy}

One can see stresses occur in the same point than before : along $\underline{e_x}$ at the circle. Moreover, the stresses are much higher at considered point, implying the expected behavior of interaction and the fact that we have quicker/bigger failure when considering several point.

This results into the fact that the stress concentration factor K_g is bigger when we have two holes. Here we have $K_g = 9.53$, compared to $K_g = 3.06$ for one hole, i.e. 3 times greater. This means that the stresses undergone by the structure are greater when there is interaction between holes ; and so the structure is more fragile.

7 Conclusion

This practical work has been the opportunity to approach the subject from a theoretical point of view and to then derive numerical results from it. We have quiet simply been obtaining results on complex mesh using symmetries, simplifying widely the structure.

Once this meshing part achieved, we have used the codes to work on structure's failure : their location and associated stresses' magnitude.

Eventually, the symmetries have once again found themselves very useful in the case of several holes. The limit of such a method (symmetry of BC) restrained the number of holes but such a method is quiet powerful when looking to obtained results.

For what is up to these practical works overall, they permitted to work in efficient ways on complex problems while looking to several different points one could look at for such structures. Thus, even though these calculations don't come together with practical manipulation and lab testing, they've been a good opening to FEA.