Universitat Polytechnica de Catalunya
MSc Computational Mechanics
Spring 2018

Computational Structural Mechanics and Dynamics

Practice 1

Due 16/03/2018

Alexander Keiser
Anurag Bhattacharjee
Luan Malikoski Vieira

Exercise 1

For the first part of this assignment a thin plate under self weight, as depicted in Figure (), will be analyzed.

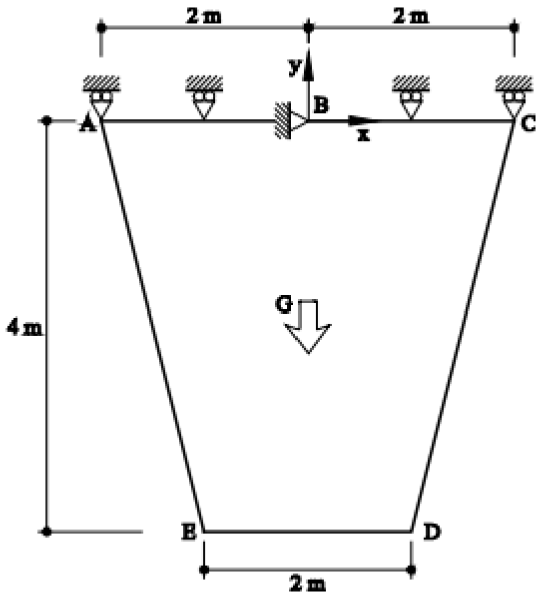


Figure 1: Thin plate under self-weight

The boundary conditions and assumptions for the problem are as follows:

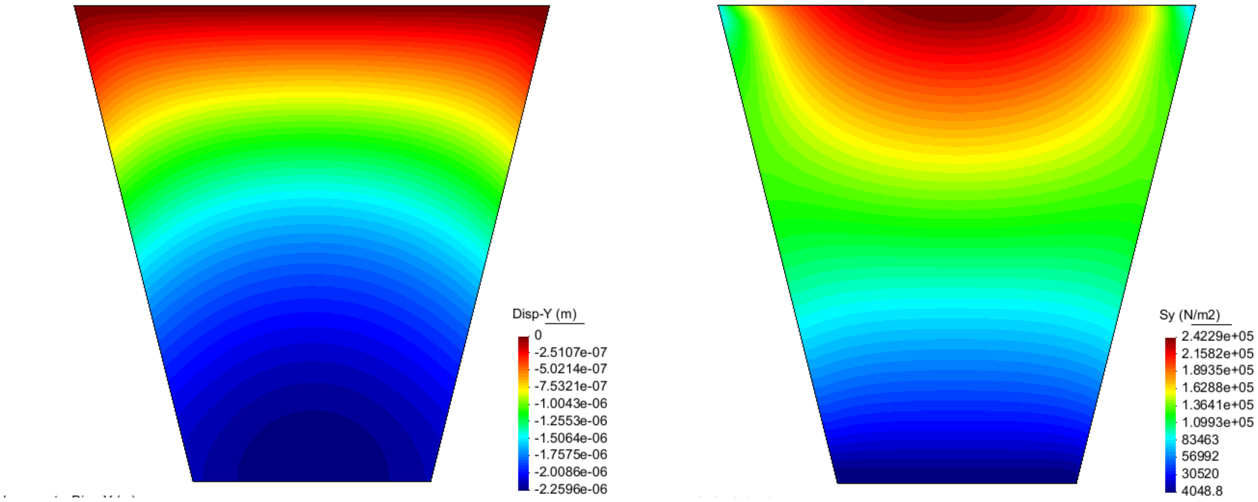
- Segment A-C: $u_y = 0$ and $u_x \neq 0$ (free).
- Point B: $u_x = u_y = 0$ (Fixed).
- Plane Stress State

The material property and problem data are summarized as follows:

Material	**E [MPa]**	$\gamma [N/m^3]$	ν [-]	**Thickness [m]**
Steel	2.1E5	68760	0.3	0.1

Table 1: Material properties and problem data

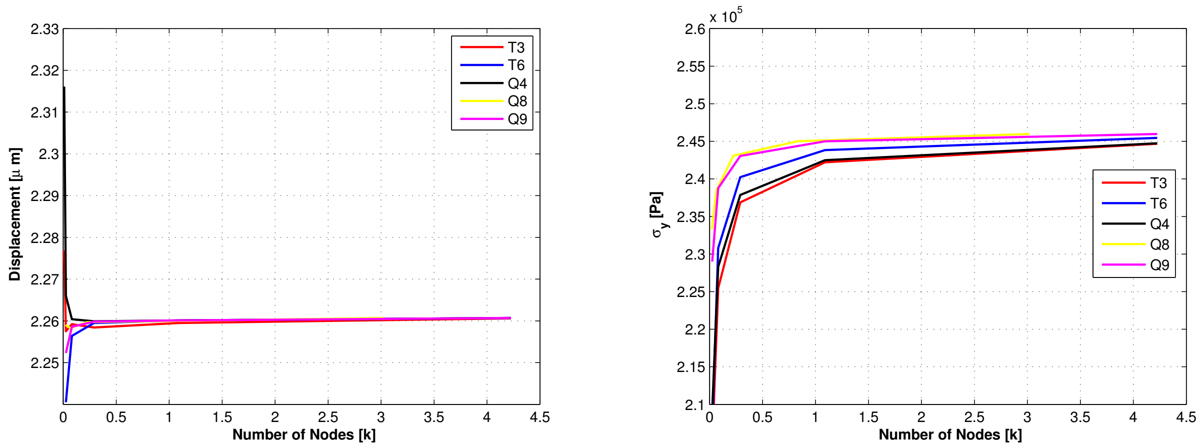
The stress in the fixed point (B in Figure ()) and displacement at the center of the side B-D is evaluated for different element type and mesh densities. The element type used are as follows: 3-Noded Triangular (T3), 6-Noded Triangular (T6), 4-Noded Quadrilateral (Q4), 8-Noded Quadrilateral (Q8) and 9-Noded Quadrilateral (Q9) elements. As a reference, analytical values for the stress at point B and displacement at center line of the thin plate free end are $\sigma_y = 0.247 GPa$ and $u_y = 2.26 \mu m$.
The stress distribution σ_y and displacement u_y on the thin plate for the case where a T3 elements mesh with 1089 nodes were used is shown in Figure (2).



(a) (b)

Figure 2: Simulation T3-1k nodes: (a) Displacement u_y; (b)σ_y Stress

Figure (3) shows the convergence of the stress σ_y and displacement u_y, at point B and middle point of segment E-D respectively, with respect to mesh nodes density.



(a) (b)

Figure 3: Mesh convergence test: (a) Displacement u_y; (b)σ_y Stress

As observed in Figure (3) (a), all mesh types are converges to a value near of the exact solution $\approx 2.26\mu m$ as the mesh density is increased. However, the rate at which each method converges is different, as expected the T3 method has a slower convergence rate as the others. When a mesh density of around 4k nodes is employed, all mesh types provide almost same solution for the displacement.

When it comes to stress evaluation at point B, Figure (3) (b) shows a clear relation between the order of elements and its accuracy with linear elements (T3 and Q4) having less accuracy and lower converge than the quadratic counterparts (T6, Q8 and Q9).

Exercise 2

In the second part of this assignment a reinforced concrete structure with two holes under stress from a load at the top,as seen in Figure (4), will be analyzed. We will also analyze additional stresses generated from sagging of the central column.

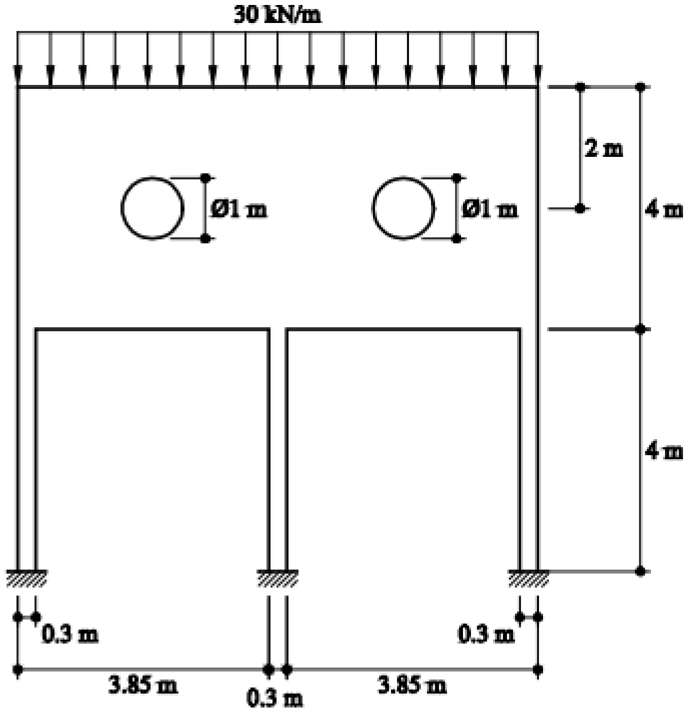


Figure 4: Reinforced concrete structure with two holes

The boundary conditions and assumptions for the problem are as follows:

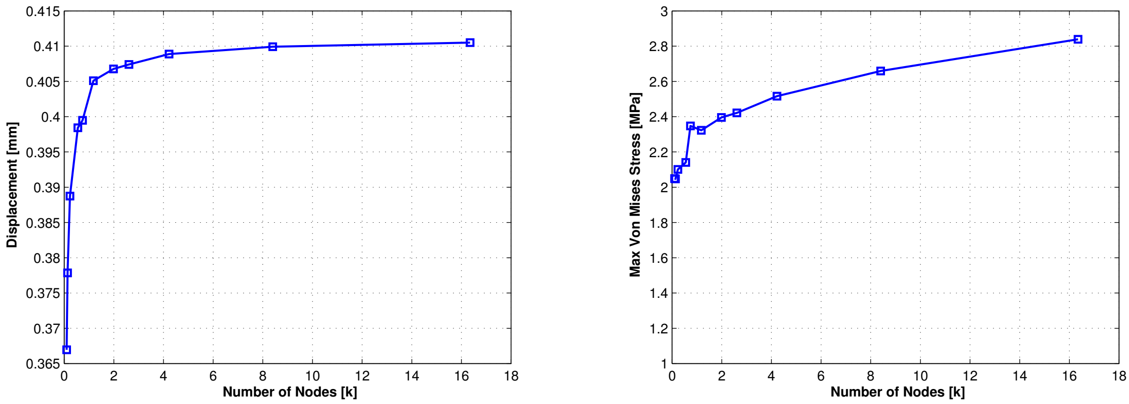
- Plat top side: Uniform distributed load (30kN/m).
- Side Columns supports: $u_x = u_y = 0$ (fixed).
- Middle Column support: $u_x = 0$ (Fixed)and $u_y \neq 0$.
- Plane Stress State

The material property and problem data are summarized as follows:

Material	**E [MPa]**	ν [-]	**Thickness [m]**
Concrete	3.0E10	0.2	0.2

Table 2: Material properties and problem data

Before starting the analysis a mesh convergence test, using T3 elements, was conducted in order to define the *"ideal"* mesh for the problem. This was done by running a the problem with the middle column free to move (no constrains). This setup was chosen considering that probably a problem with free middle column would have a lower convergence rate than the same problem with this column fixed (constrained). This way, the found converged mesh would be dense enough for all possible displacement setups.
As it can be seen in Figure (5), the maximum displacement in y direction (middle point) and maximum Von-Mises stress (Located at side columns corner) were tracked. Convergence for displacement were reached for around 8k nodes, while stress convergence could not be achieved for the maximum number of nodes (16k) used. The stress convergence could be achieved if a mesh study were carried-out with local refinement in high stress concentration regions, however this is not the scope of this report.



(a) (b)

Figure 5: Mesh convergence test: (a) Maximum Displacement; (b)Maximum Von-Mises Stress.

In order to keep a feasible computational cost, the mesh with 8k nodes, shown in Figure (6) was chosen as the *"ideal"* one.

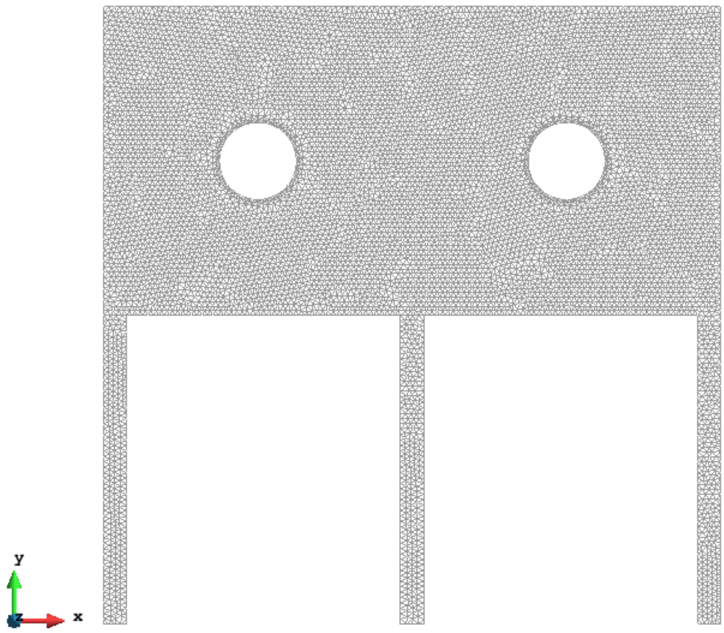


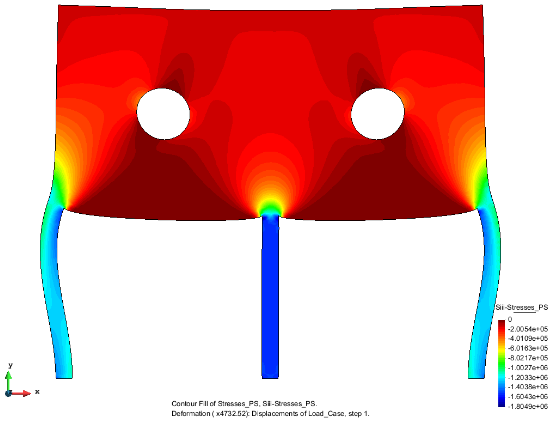
Figure 6: Final mesh: 8k nodes T3.

Now that an appropriate mesh size is obtained, the analysis for different prescribed displacements (δ) of the center column can be carried out. The analysis will consider 5 displacements (δ) condition as follows: 0.0 mm (constrained), 0.1mm, 0.2mm, 0.3mm and free case (solved in the mesh convergence test).
As the structure is made of **Concrete**, which is a **Brittle** material, for failure point of view analysis, Von-Mises stress can not be taken into account. In this case, principal stresses ((σ_1, σ_2 and σ_3) are the appropriate ones which are usually employed in failure theory criteria of Brittle materials (e.g.,Maximum Normal Stress, Coulomb-Mohr or Christensen) and compared with maximum tensile (σ_t) and compressive (σ_c) material strength.
The results for the three principal stresses is shown in Table (3). As expected, the maximum principal stress is a compressive one. Also for a reference comparison, the compressive strength of construction concrete is within the range of 15MPa to 30MPa, which is much higher than the maximum values found. Thus, by the Maximum Normal Stress theory, if the structure is made of only pure concrete, it should not fail under this load and displacements conditions.

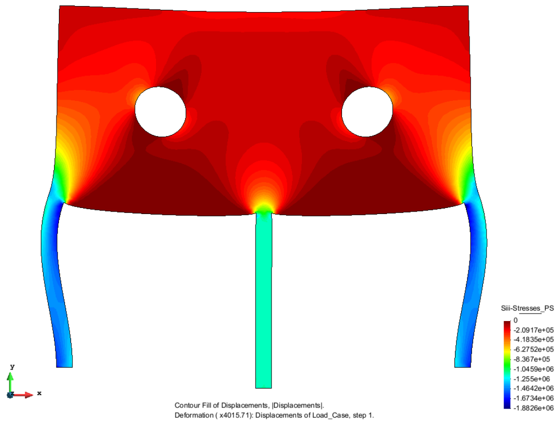
	0.0	0.1	0.2	0.3	free
σ_1 [Pa]	5.07E+05	5.79E+05	6.52E+05	7.25E+05	7.97E+05
σ_2 [Pa]	-3.41E+05	-3.15E+05	-3.58E+05	-4.01E+05	-4.44E+05
σ_3 [Pa]	-1.80E+06	-1.88E+06	-2.14E+06	-2.40E+06	-2.66E+06

Table 3: Maximum principal stresses for each displacement boundary condition

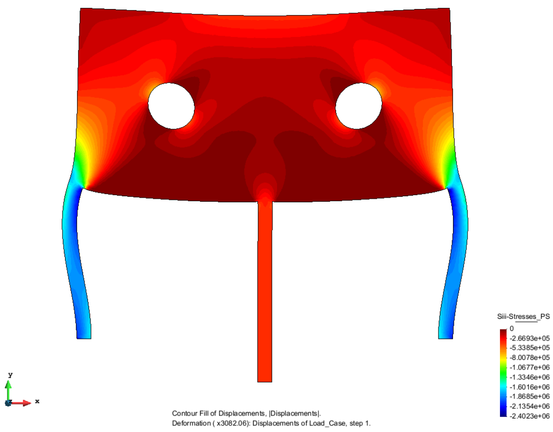
It is also noticed in Table (1) that the higher the displacement of the center column, the higher are the maximum principal stresses. The position of those maximum stress principal stress σ_3 can be seen in Figure (7). It can be noticed that as the middle column is released from constrained condition, the side columns get more loaded. As a result apart of being increased, the position of maximum stress in the structure is shifted from the middle bar ($\delta = 0$) to the two side bar as seen in the Figure (7) (a) to (d) .



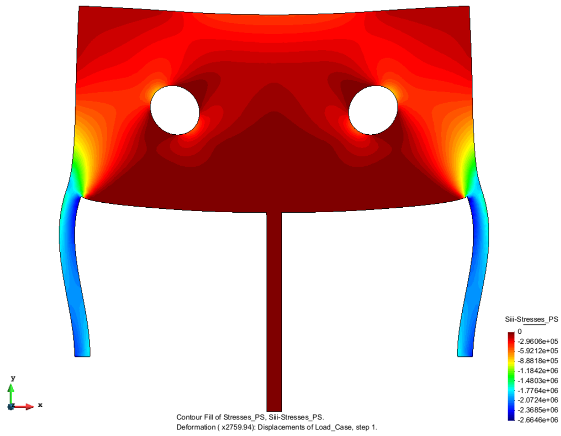
(a) $\delta = 0$



(b) $\delta = 0.1mm$



(c) $\delta = 0.3mm$



(d) Free

Figure 7: Maximum Principal Stress (σ_3)

.

Exercise 3

In the third part of this assignment the reinforced concrete structure with simple supports, seen in Figure (), meshed with 4-noded quadrilateral elements (Q4) is analyzed.

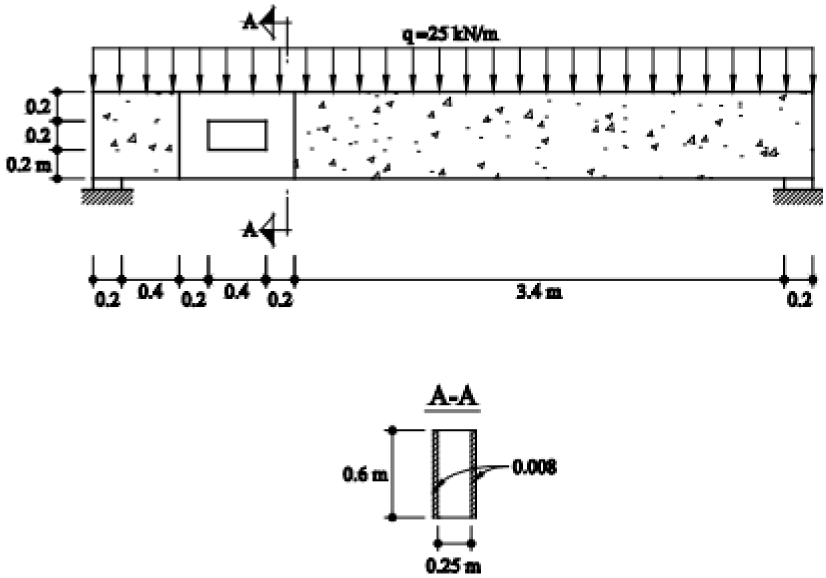


Figure 8: Reinforced Concrete Structure with simple supports

The boundary conditions and assumptions for the problem are as follows:

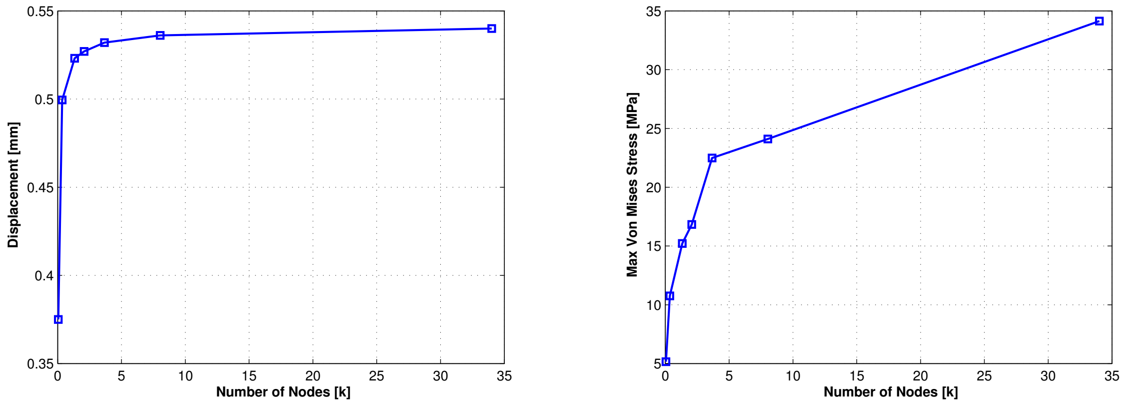
- Plat top side: Uniform distributed load ($q = 25kN/m$).
- Side supports: $u_x = u_y = 0$ (fixed).
- Plane Stress state.

The material property and problem data are summarized as follows:

Material	**E [MPa]**	ν [-]	**Thickness [m]**
Concrete	3.0E10	0.2	0.25
Steel	2.1E11	0.3	0.016

Table 4: Material properties and problem data

A mesh study was carried out in order to define a converged mesh that will be used in the subsequent analysis. As it can be seen, in Figure (9) (a), the maximum displacement, at the center of the plate, convergence is attained for a 8k nodes mesh. However, the maximum Von-Mises, measured at the reinforcement plate inner corner, do not converge for the considered mesh densities. Again, a mesh refinement focusing on refinement in high-stress concentration regions could lead to a converged mesh with lower density than those considered here.



(a) (b)

Figure 9: Mesh convergence test: (a) Maximum Displacement; (b)Maximum Von-Mises Stress.

Even though no convergence was attained for the stresses, the mesh with 8k nodes was chosen as the "ideal". The chosen mesh is shown in Figure ().

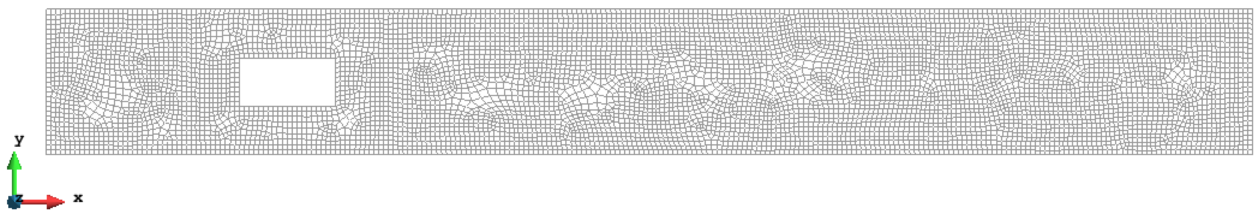
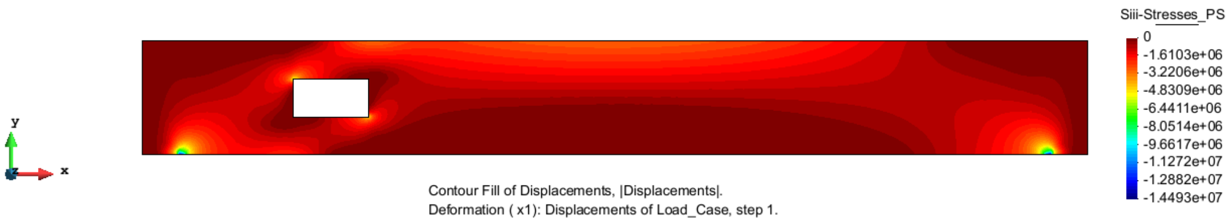


Figure 10: Final mesh: 8k nodes Q4

In order to evaluate the effect of adding a steel plate to reinforce the concrete structure, a simulation with the "ideal" mesh were run for a case where no reinforcement steel plate is present. The third principal stress (σ_3) for each case is shown in Figure (11)

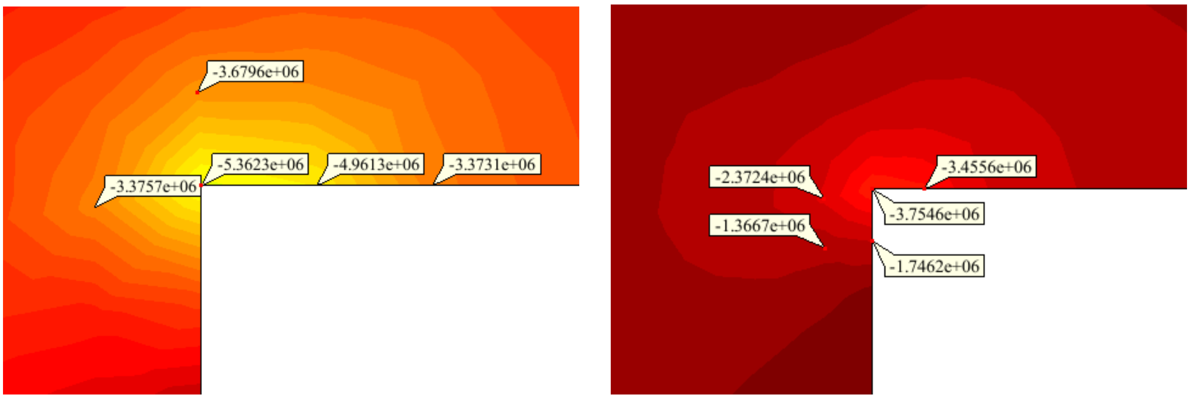


(a)

(b)

Figure 11: Maximum principal stress (σ_3): (a) Original Structure; (b) Reinforced Structure

A clear reduction of the stresses in the region of the ventilation hole is observed when the two 0.008m (Modelled as one plate of 0.016m of thickness) thickness plates are placed in each face of the concrete plate. In Figure (12), a detail view, for each case, in the region of the hole corner is provided.



(a) (b)

Figure 12: Maximum principal stress (σ_3): (a) Original Structure; (b) Reinforced Structure

Here it can be noticed that the maximum σ_3 stress, which is located at the corner stress concentration point, is reduced from around 5.4MPa to 3.75MPa.

Exercise 4

In the last part of this assignment, a prismatic water tank, see in Figure (), made of reinforced concrete under hydro-static load is analyzed.

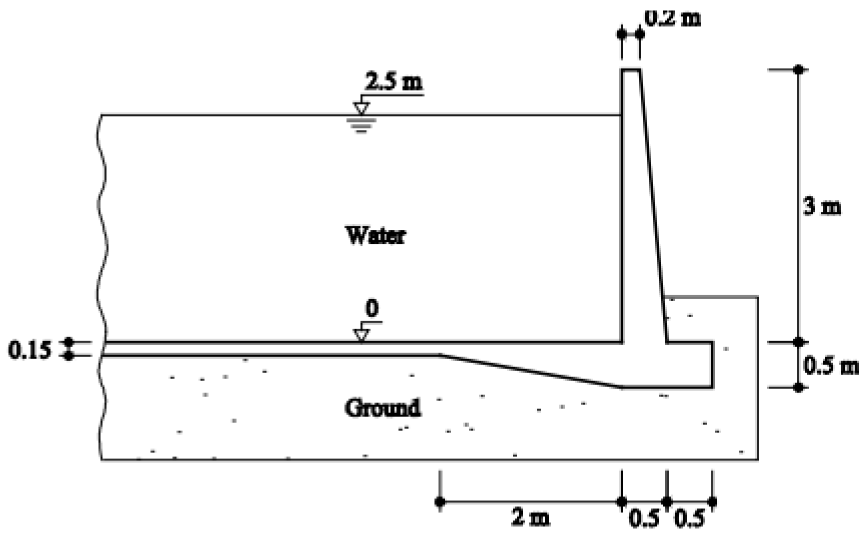


Figure 13: Prismatic water tank

The boundary conditions and assumptions for the problem are as follows:

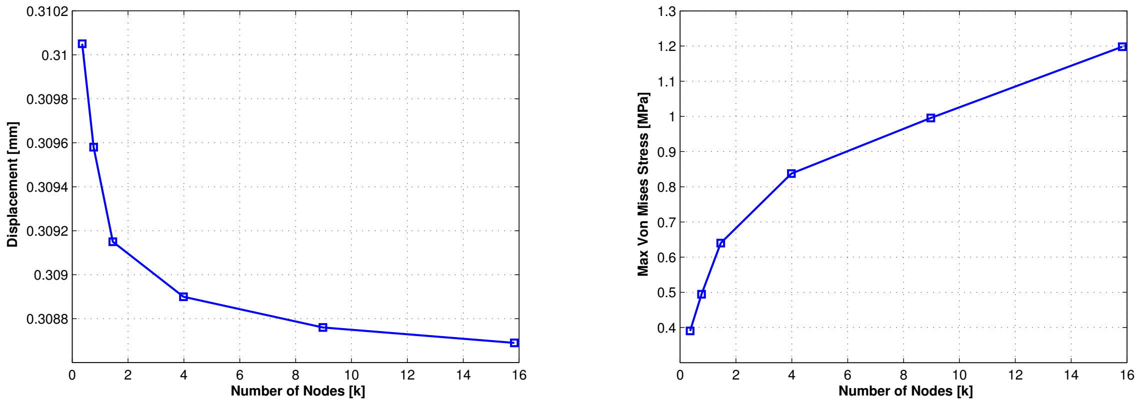
- Tank Bottom inner face: Uniform distributed load due to Hydro-static pressure ($\rho gh = 24451 N/m^2$).
- Tank wall inner face: Linear distributed load due to Hydro-static pressure (from $\rho gh = 24451 N/m^2$ at bottom to zero at top).
- Tank bottom far left end: $u_x = 0$ (Fixed) and $u_y \neq 0$ (free).
- Tank Bottom outer surface: Elastic constrain ($K_y = 50E6 N/m^3$).
- Tank Bottom outer surface (vertical surface in contact with soil): Elastic constrain ($K_x = 50E6 N/m^3$)
- Plane strain state (thickness of wall normalized to 1m)

The assumption of a elastic constrain ($K_x = 50E6 N/m^3$) in the lateral (vertical) contact between soil and dam was made considering that due to the load condition (+x direction), the soil is being compressed by the dam structure in this direction, as it is happening also for the bottom soil in +y direction. The material property and problem data are summarized as follows:

Material	**E [MPa]**	$\gamma [N/m^3]$	ν [-]
Concrete	3.0E10	24000	0.2

Table 5: Material properties and problem data

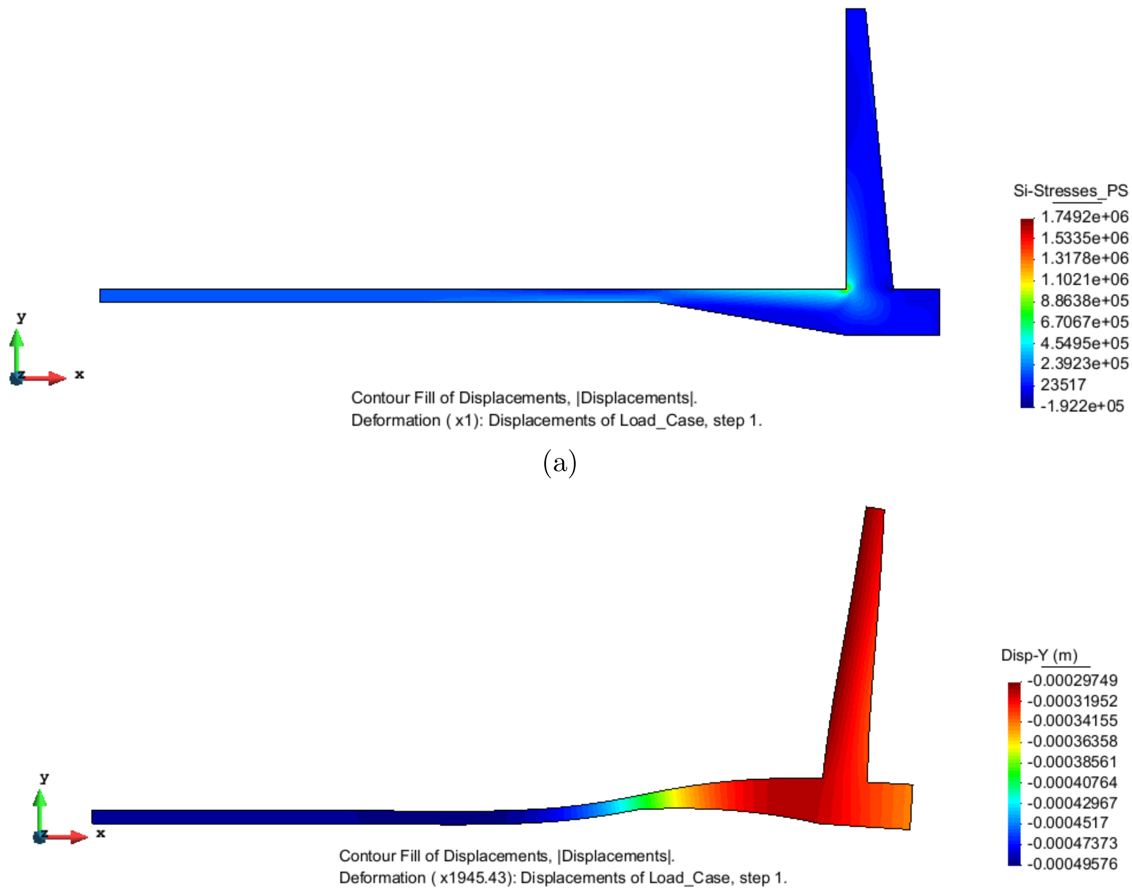
As did for previous problems, before drawing conclusions towards the stress state of the structure, a mesh convergence test was carried out. Displacement u_y and Von-Mises stress in the water tank inner corner (junction of vertical wall and tank bottom) was tracked. Results are shown in Figure (14).



(a) (b)

Figure 14: Mesh convergence test: (a) Maximum Displacement; (b)Maximum Von-Mises Stress.

As it can been, both displacement and Von-Mises stress did not have full convergence. Due to limitations to run the problem with more dense meshes. Thus, results will be analyzed in a qualitative way using the most dense mesh obtained (16k nodes).



(b)

Figure 15: (a) Displacement u_y and deformed shape;(b)Maximum principal Stress

From Figure (15) it is observed the maximum principal stress (σ_1) in the tank wall as well as the deformed shape and displacement in y direction. It can be seen, as expected, due to the bending moment caused by the hydro-static load, the maximum stress occur in the inner corner of the tank. A detail view of the stress concentration region is shown in Figure ().

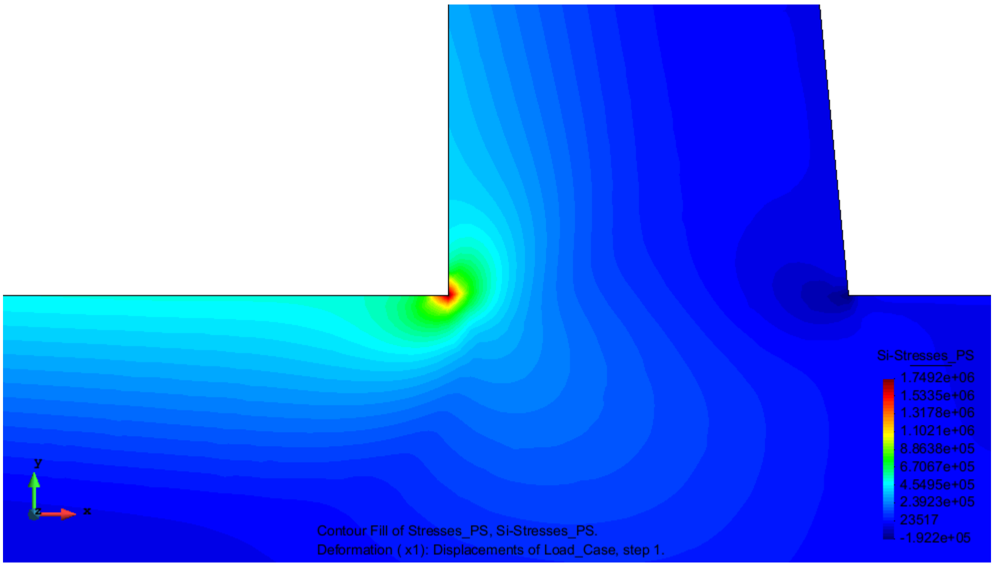


Figure 16: Stress concentration in tank inner corner