UNIVERSITAT POLITÈCNICA DE CATALUNYA
MSc IN COMPUTATIONAL MECHANICS/NUMERICAL METHODS IN ENGINEERING
COMPUTER STRUCTURAL MECHANICS AND DYNAMICS

# GiD Assignment 1

Krupesh B. Shivaprakash
Abhishek Tanwar
Gabriel Valdés Alonzo

March 9th, 2017

## Problem 1

To analyze the problem, geometry of the thin plate was drawn in GID as per given dimensions. Since it is given that the plate is thin, we can infer that the posed problem is of planar deformation.

The mesh was structured using triangular and quadrilateral elements:

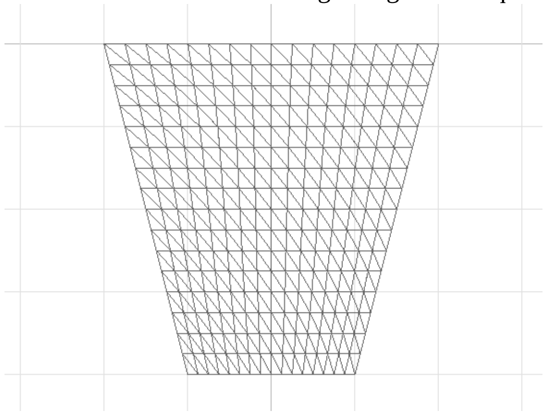


Figure 1.1(a): Triangular element mesh

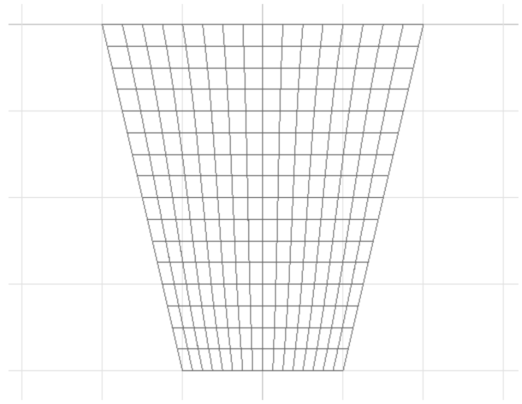


Figure 1.1(b): Quadrilateral element mesh

After applying given boundary conditions, material properties and loads (self-weight in this case) different solutions were analyzed by refining the triangular element mesh with 3 and 6 nodes and quadrilaterals with 4, 8 and 9 nodes.

Triangular element normal mesh (i.e. with linear shape functions) with 3 nodes:

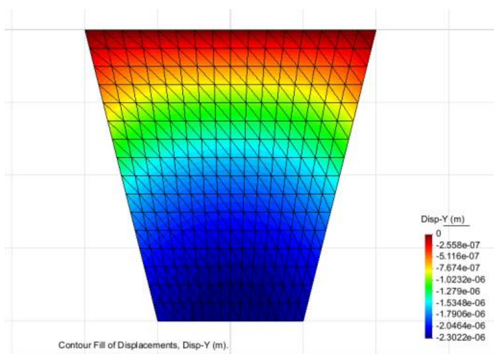



Figure 1.2(a): Displacement Variation

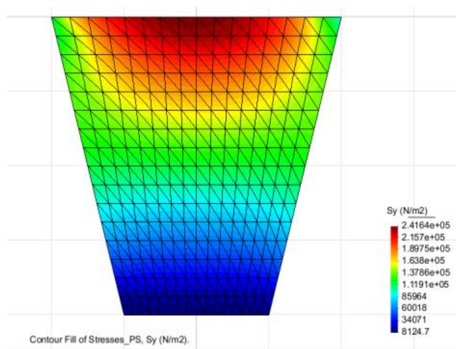



Figure 1.2(b): Stress ($\sigma_y$) variations

Triangular element quadratic mesh with 6 nodes:

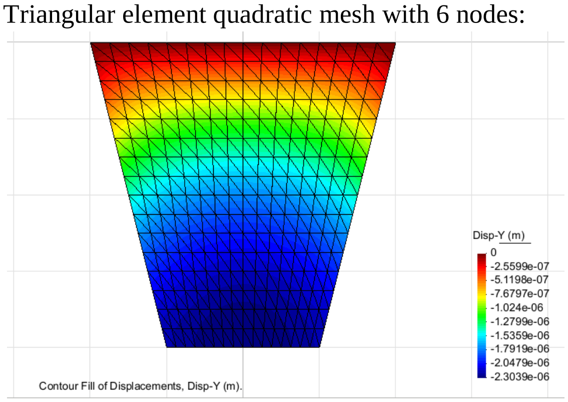



Figure 1.3(a): Displacement Variation

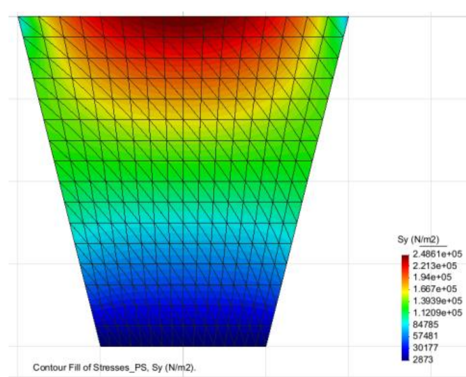



Figure 1.3(b): Stress ($\sigma_y$) variations

Quadrilateral element normal mesh with 4 nodes:

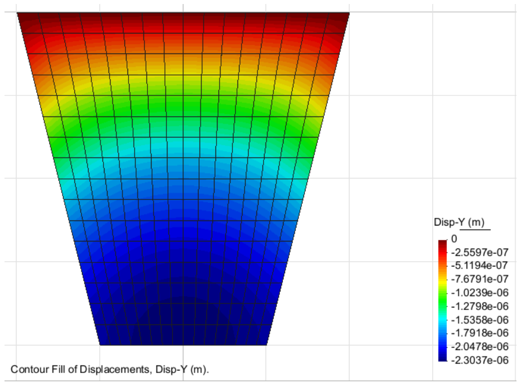



Figure 1.4(a): Displacement Variation

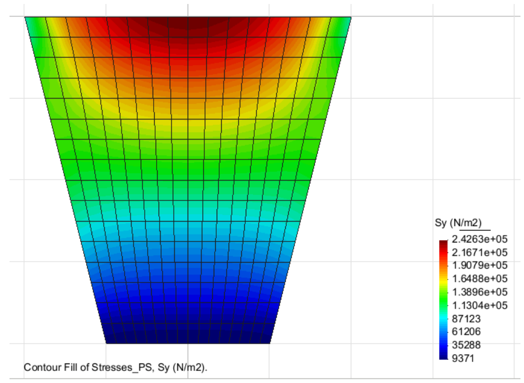



Figure 1.4(b): Stress ($\sigma_y$) variations

Quadrilateral element quadratic mesh with 8 nodes:

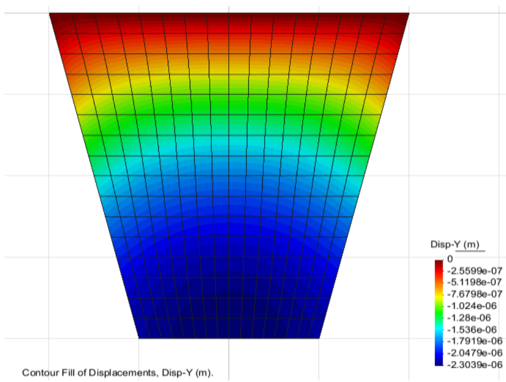



Figure 1.5(a): Displacement Variation

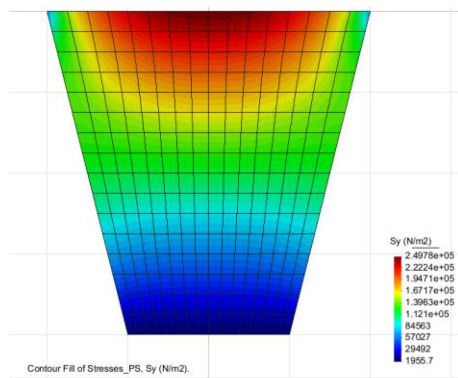



Figure 1.5(b): Stress ($\sigma_y$) variations

Quadrilateral element quadratic9 mesh with 9 nodes:

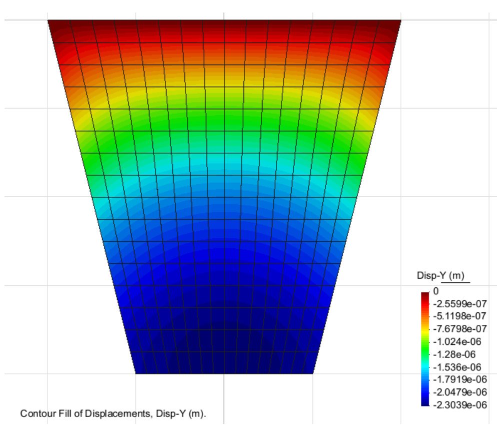



Figure 1.6(a): Displacement Variation

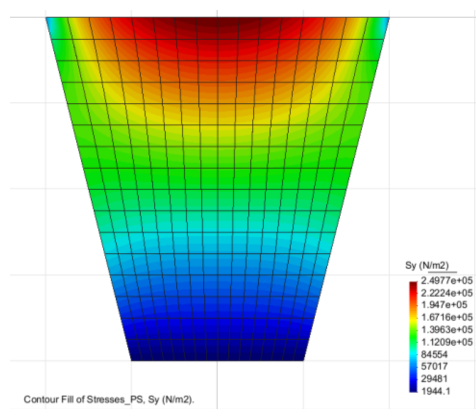



Figure 1.6(b): Stress ($\sigma_y$) variations

The sought solution given as per problem is:
Displacement y at center of side ED = 2.26e-6 m
Stress at point B ($\sigma_y$) = 0.247 $MN/m^2$

Table 1.1: comparison of results

| Element Type | Number of elements | Degree of Freedom | Stress ($\sigma_y$) at point B {$MN/m^2$} | Displacement at (y) center of ED {m} | Relative error of $\sigma_y$ (%) | Relative error of displacement (%) |
|---|---|---|---|---|---|---|
| Triangle with 3 nodes | 512 | 289 | 0.24164 | 2.1022e-6 | 2.170 | 6.980 |
| Triangle with 6 nodes | 512 | 1089 | 0.24776 | 2.2337e-6 | 0.307 | 1.160 |
| Quadrilateral with 4 nodes | 256 | 289 | 0.24263 | 2.255e-6 | 1.769 | 2.210 |
| Quadrilateral with 8 nodes | 256 | 833 | 0.24578 | 2.2599e-6 | 0.494 | 4.42 |
| Quadrilateral with 9 nodes | 256 | 1089 | 0.24579 | 2.2599e-6 | 0.489 | 4.42 |

If we observe displacement variation then we can see that there is no displacement at top of plate and as we move down towards end ED the displacement increases, which is a clear indication of the effects of self-weight acting on the plate. Similarly if we see stress variation then it is maximum at point B which is constrained and experience the most stress due to self-weight of the plate. On comparison of the results with solution it was observed that quadrilateral elements gave a better result than triangular elements. As the mesh was refined further the result approaches closer to sought solution.

**Problem 2**

The structure proposed is a very thin one ($0.2\ m$) which is computed in plane stress assumption, so it is drawn in 2D using GiD.
The three pillars that support the structure are fixed in both directions, considering that the middle one is displaced from its intended position in a quantity $\delta$. Since there is not a fixed value for the displacement, three values will be considered to study the progress of the stresses during sinking. The structure is then analyzed for the cases where $\delta = 0.001\ m$, $\delta = 0.01\ m$ and $\delta = 0.1\ m$. Self-weight is always considered with a factor of 1.
The force considered is the linear force on the top, which is inputted without any modification in the model.
The system after applying all conditions can be seen in figure 2.1:

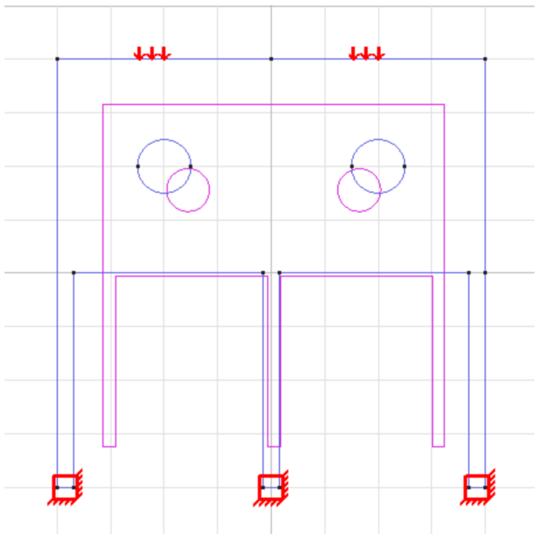


Figure 2.1: conditions applied on the structure.

A structured triangular mesh is considered for the problem. Since it's asked for 3-noded triangular elements, linear elements are considered for the computation. The mesh is shown in figure 2.2:

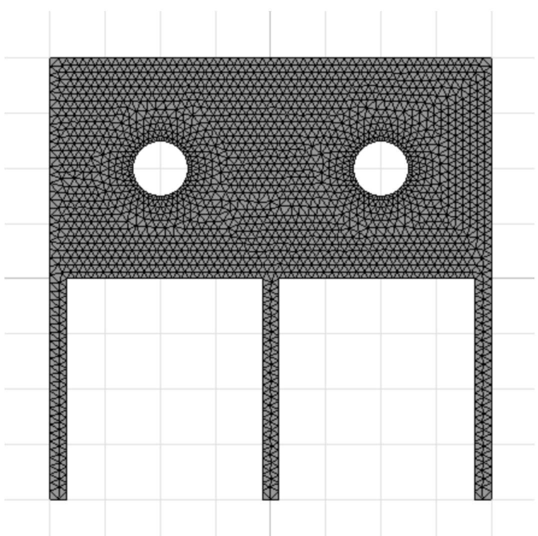
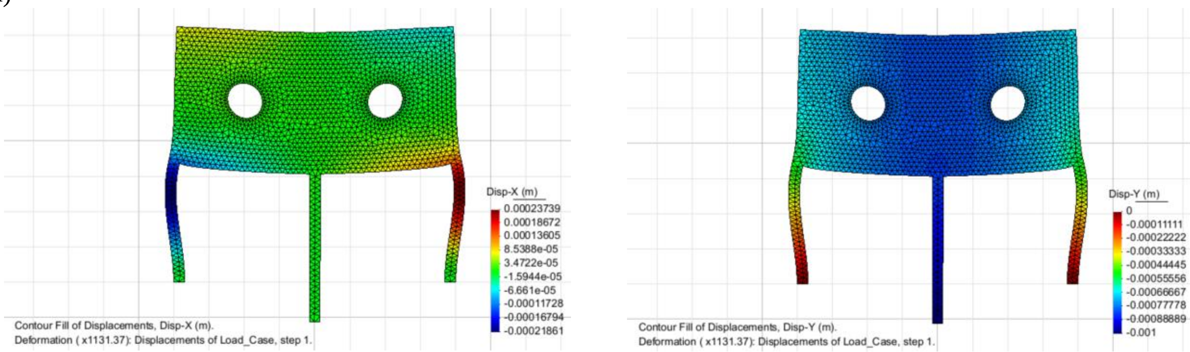


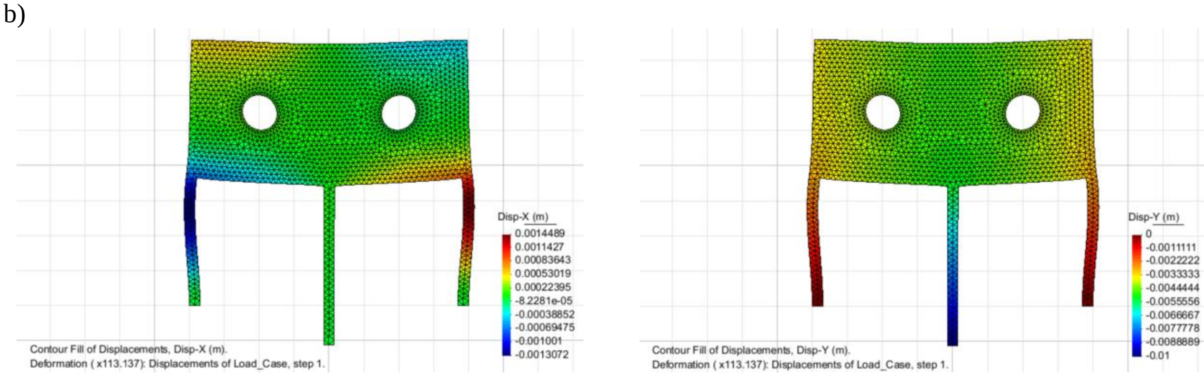
Figure 2.2: triangular element mesh.

With everything set now the structure now is computed for the three proposed values of $\delta$. The deformations are shown in the figure below:

a)




b)




c)

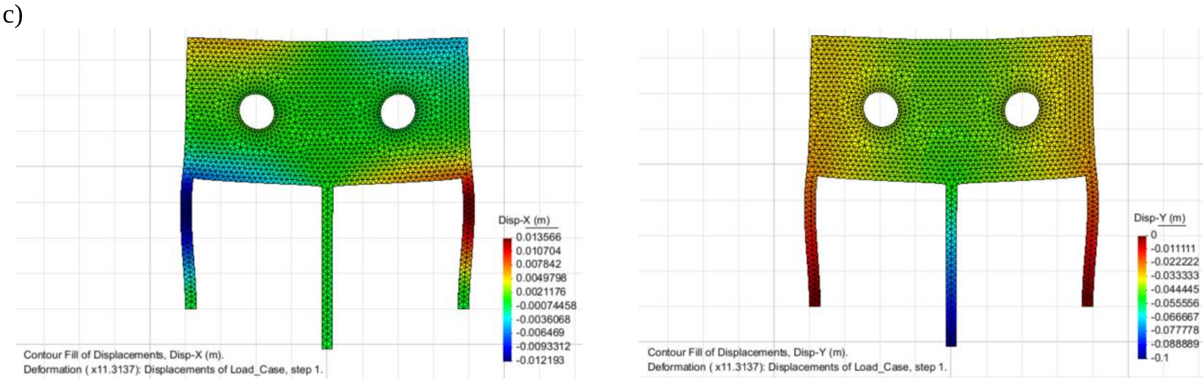
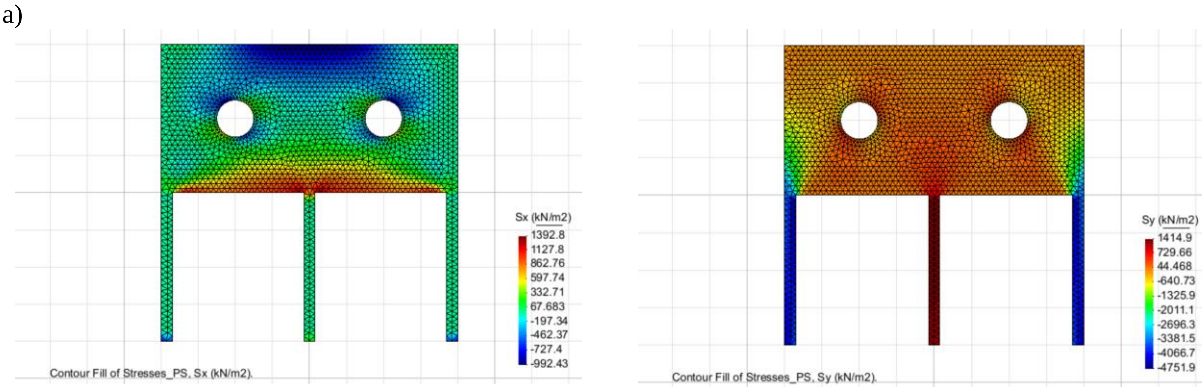


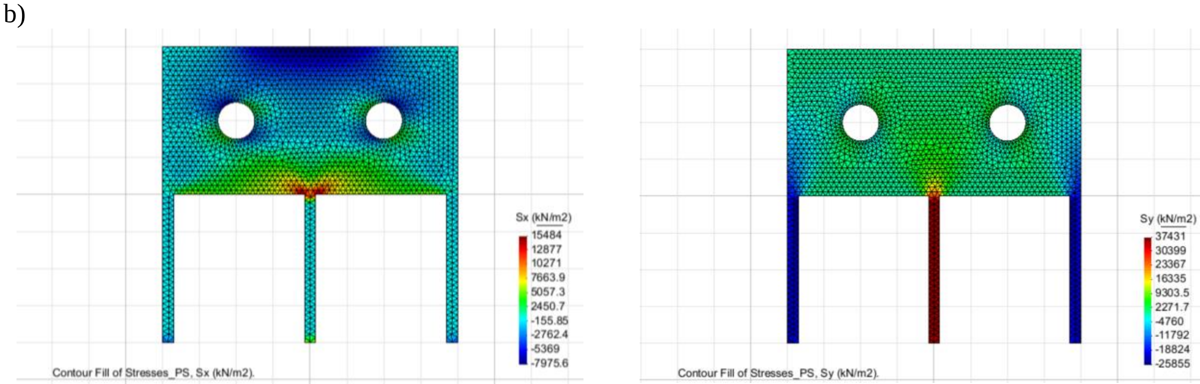

Figure 2.3: a) displacements for δ=0.001 m; b) displacements for δ=0.01 m; c) displacements for δ=0.1 m.

And the stresses are shown in the next figure:

a)



b)




c)

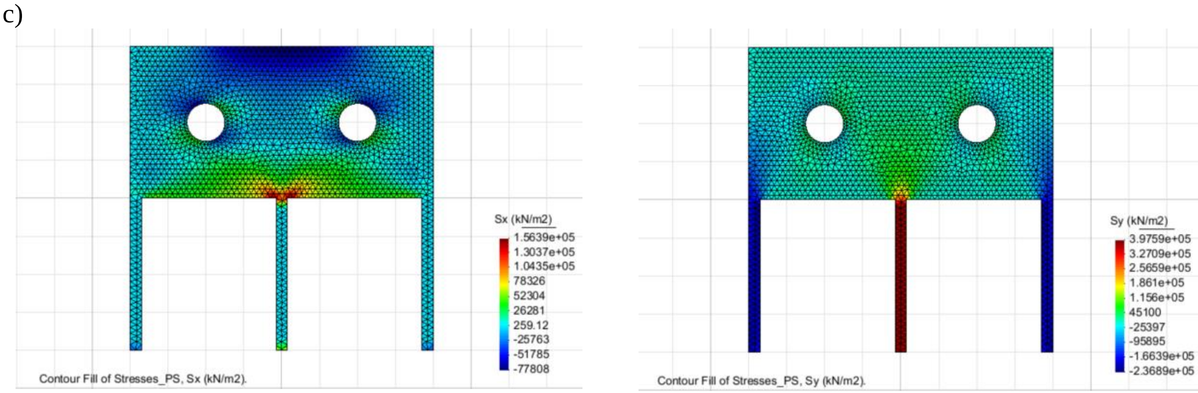



Figure 2.4: a) stresses for δ=0.001 m; b) stresses for δ=0.01 m; c) stresses for δ=0.1 m.

As can be seen from the images, there is not a clear relationship between the value of the deformation and the value of the stresses generated by it. What can be seen clearly is the tendency of how the stresses behave, that is a concentration of stresses in the union of the central pillar with the structure. This is due to the fact that the central pillar is acting in a way similar to that of the punching stress, in which only the vicinity of the column is affected to the problem. As for the other direction, it can be seen that the stresses tend to concentrate on the central pillar more when the deformation increases, creating a stronger gradient in every step.

Images show also the presence of bending due to the descent of the central column. As the stresses concentrate on the border of the central column, the top border of the structure directly above it suffers from stresses in the opposite direction, which correlates with the deformation experienced and visible on figure 2.3.

**Problem 3**

The system of concrete plate with ventilation hole is drawn using GiD. This system is analyzed under two conditions: with and with-out reinforced steel plates. The geometry with boundary conditions is shown in Figure 3.1. There is uniform load of 25 kN/m on the top of the plate. The system is simply supported as shown in the figure.

Figure 3.1 Diagram of Geometry of Concrete plate with Boundary Conditions

Mesh diagram is shown in figure 3.2. Quadrilateral elements with four nodes are considered for meshing. In the case of the concrete plate with reinforced Steel the nodes are collapsed in the interface to have a joint displacement.

Figure 3.2 Diagram of Mesh

Owing small thickness of plates, plain stress hypothesis is assumed. The displacement in y direction of the concrete plate with reinforced steel is shown in figure 3.3. Maximum displacement is observed in the center portion of the plate. Minimum displacement observed in fixed ends and its surrounding area. Small vertical displacement are observed near the ventilation hole.

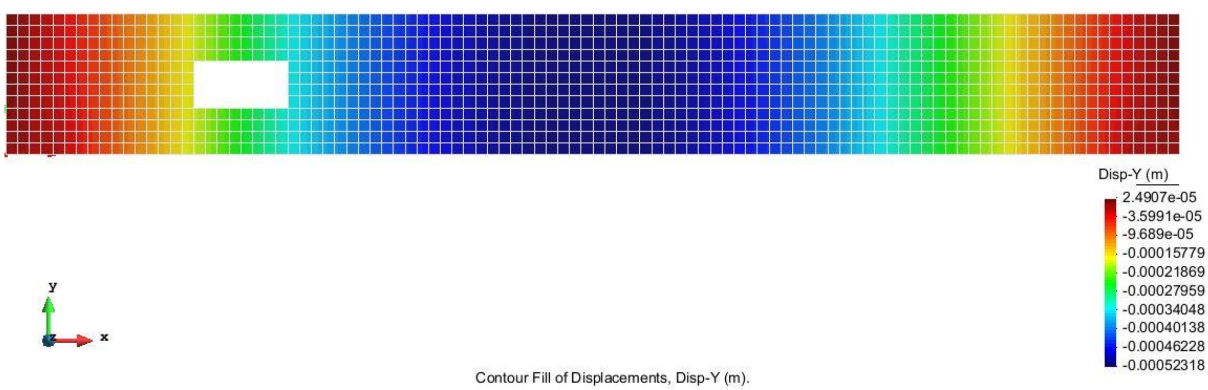



Figure 3.3 Diagram of Displacements y of Concrete Plate with Reinforced Steel sheets.

The Vertical stresses are shown in the Figure 3.4 and Horizontal stresses are in Figure 3.6. Concentrated stresses are observed in the corners of ventilation hole and also maximum stresses are observed in this area. The reinforced steel sheets help overcome failure of the plate in these areas, effect that can be seen on the figures below.

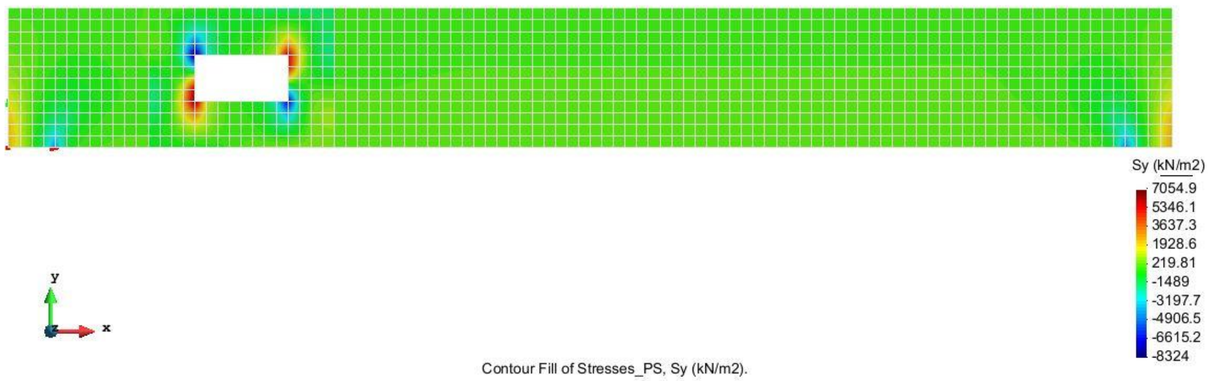



Figure 3.4 Diagram of Stress in y direction of Concrete Plate with Reinforced Steel Plates

Owing to non-symmetry, more horizontal stresses are developed near ventilation hole pipe as shown in figure 3.5.

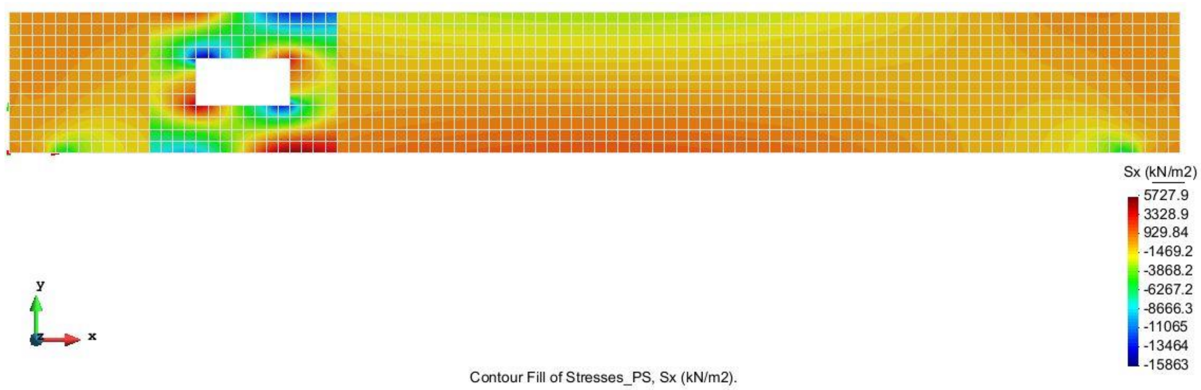



Figure 3.5 Diagram of Stress in x direction of Concrete Plate with Reinforced Steel sheets.

The system was also analyzed without using reinforced steel sheets. Higher vertical displacements are observed which in the general case lead to failure of the system, hence the reinforcement of steel sheets is a necessary improvement in the design, because takes part of the stress associated to the bending, as well as a mean of preventing the propagation of cracks and damages in the beam.

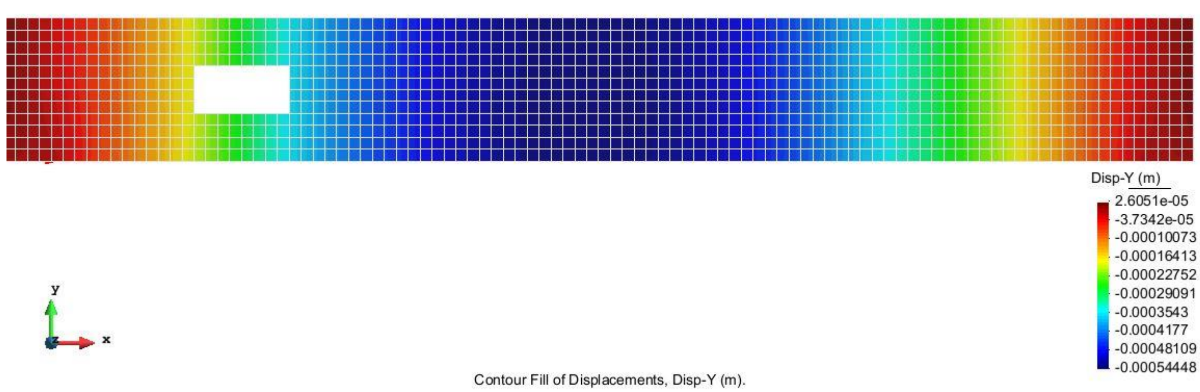



Figure 3.6 Diagram of Displacements y of Concrete Plate with-out Reinforced Steel sheets.

The stresses of concrete plate without using the reinforced steel sheets are shown in the figures below. It can be observed that more stresses appear near the fixed support as well.


Figure 3.7 Diagram of Stress in y direction of Concrete Plate with-out Reinforced Steel sheets.

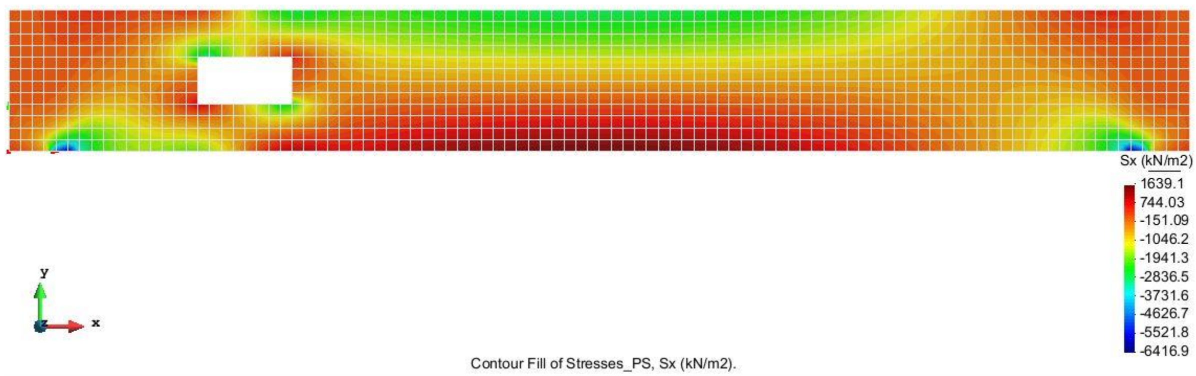



Figure 3.8 Diagram of Stress in x direction of Concrete Plate with-out Reinforced Steel sheets.

**Problem 4**

For analyzing the structure 2D slice of the structure is drawn in GiD. Since we are considering a planar deformation, we consider a unit thickness ($t = 1$ m), which allows us to have referential stresses and forces for any length that the wall has.

The floor of the tank is infinitely long, so up to 5 meters from the wall are considered in order to avoid any stress concentration near the corner, and a symmetry condition is applied to express the infinite extension (the floor is restricted to move in X but has allowed displacement in Y). In this case the ground works as an elastic support, so the value is inputted as shown, considering the thickness of the slice of the dam (which translates the load coefficient to $50\ \mathrm{MN/m^3}$). The passive effect of the ground on the wall is not considered because we don't have dimension values, and also because it is a negligible effect on the general problem. Self-weight is considered in the problem with a factor of 1.

The forces on the dam come from the pressure of the water, which is considered as linear on the wall due to the distribution $q(z) = \gamma_w z$, which gives a bottom pressure of $24.5\ \mathrm{kN/m^2}$. This is also applied uniformly on the bottom of the tank, since pressure acts in all directions. Since we consider a unit thickness, the pressure is the same as the linear force applied in the wall and floor of the tank ($24.5\ \mathrm{kN/m}$).

The structure then looks like it's shown in figure 4.1:

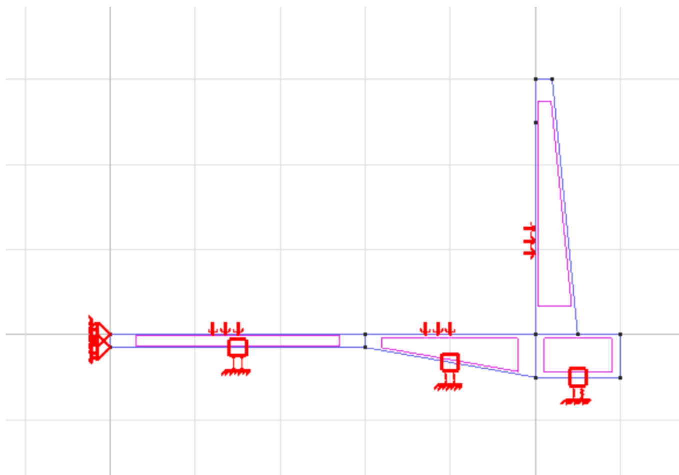


Figure 4.1: conditions applied on the structure.

A structured quadrilateral element mesh is generated for the problem, which is shown in figure 4.2:

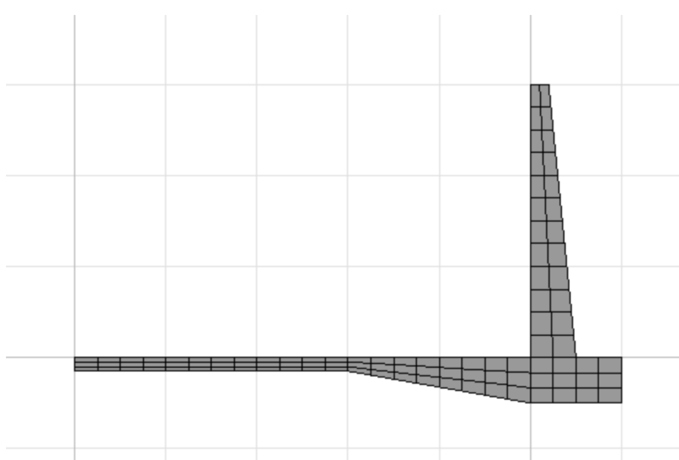


Figure 4.2: quadrilateral element mesh.

So with everything set, the structure is computed for linear, quadratic and quadratic9 elements. To check if the forces and supports are correctly inputted, the deformed state of the structure is checked, as can be seen in Figure 4.3:

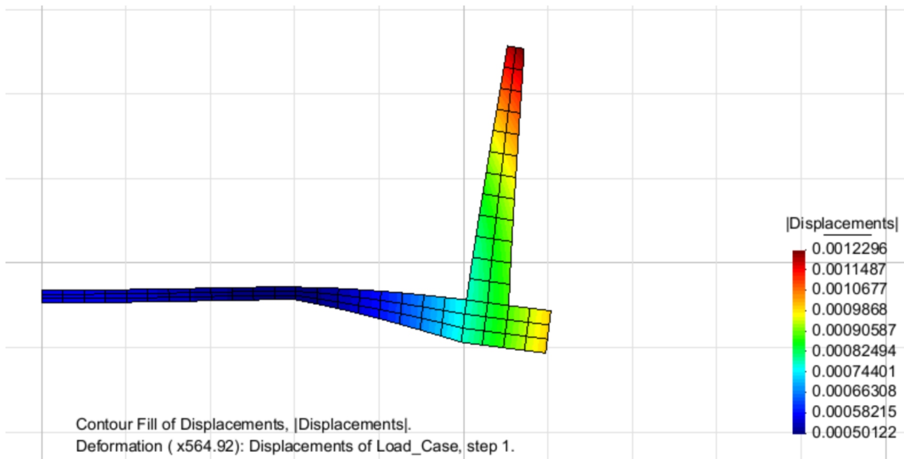


Figure 4.3: deformed state of the structure.

Which makes sense given that the wall behaves like a cantilever beam, and the floor as a slab under constant loading. Since everything is correct, the stresses for the different meshes are shown in Figure 4.4a to c:

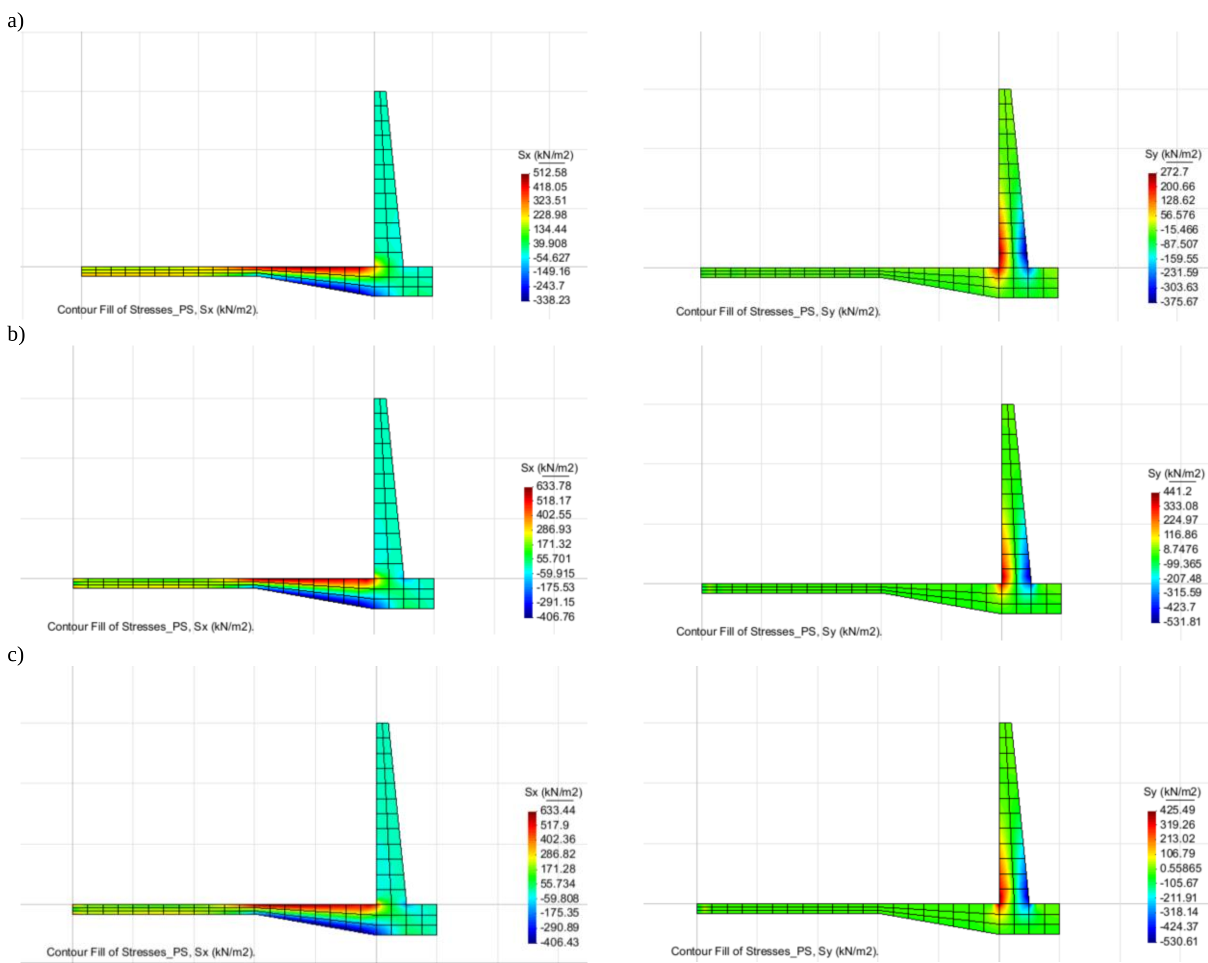


Figure 4.4: a) Stresses for linear elements; b) Stresses for quadratic elements; c) Stresses for quadratic elements with 9 nodes.

Table 4.1: comparison of results

| Type of Element | Maximum Stress Sx (kN/m$^2$) | Maximum Stress Sy (kN/m$^2$) | Minimum Stress Sx (kN/m$^2$) | Minimum Stress Sy (kN/m$^2$) |
|---|---|---|---|---|
| Linear | 512.58 | 272.70 | -338.23 | -375.67 |
| Quadratic | 633.78 | 441.20 | -406.76 | -531.81 |
| Quadratic9 | 633.44 | 425.49 | -406.43 | -530.61 |

As can be seen from the images there is no much difference between both quadratic methods, but the difference is significant between linear and quadratic. First we can see that the expansion of the higher stresses (i.e. the size of the red and blue spots) is a little bigger in the linear element, which implies that the gradient of stresses is a little lower than quadratic elements. Numerically, we have that for the X direction the stresses have a ~20% increase in the quadratic case, as for the Y direction this increase is around 50%. This can be explained due to the fact that accuracy for both kind of elements is different. It is well known that linear elements are not capable of capturing every aspect of the model of the behavior of the structure, like bending or non-linear problems, leading to an approximation error that can propagate and create the differences experienced between meshes. The fact that both quadratic give similar results lead to the conclusion that the higher order is indeed a correct solution.

For the distribution of stresses we see that it makes sense. This is because, as said before, the wall behaves as a cantilever beam, so the concentration of stresses should occur in the base of the wall, with tension stresses at the left and compression stresses at the right, which are caused by the deformation shown in 4.3. As for the floor, we have again tensional stresses on the top and compression on the bottom part, due to the way that the floor is bending, caused by the forces on the wall as well as the reaction of the soil, which is behaving in a rather flexible way.