



UPC - BARCELONA TECH  
MSc COMPUTATIONAL MECHANICS  
Spring 2018

---

# Computational Solid Mechanics & Dynamics

---

## **GiD HOMEWORK 1**

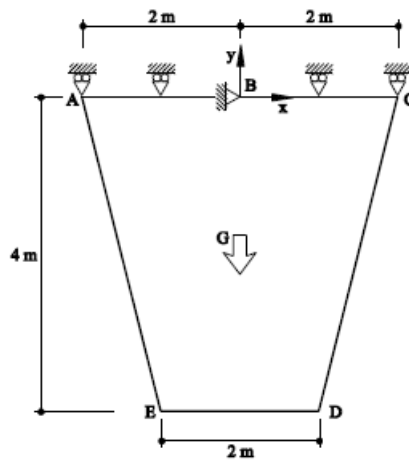
Due 16/03/2018

Prasad ADHAV

# Contents

<b>1</b>	<b>Thin plate under dead weight</b>	<b>2</b>
1.1	Purpose of the example . . . . .	2
1.2	Analysis . . . . .	2
1.2.1	Preprocessing . . . . .	2
1.2.2	Processing . . . . .	4
1.2.3	Postprocessing . . . . .	4
1.2.4	Analysis of results . . . . .	12
<b>2</b>	<b>Plate with two sections.</b>	<b>14</b>
2.1	Purpose of the example . . . . .	14
2.2	Analysis . . . . .	14
2.2.1	Preprocessing . . . . .	14
2.2.2	Processing . . . . .	17
2.2.3	Post-processing and analysis of the results . . . . .	17
<b>3</b>	<b>Plate with ventilation hole</b>	<b>23</b>
3.1	Purpose of the example . . . . .	23
3.2	Analysis . . . . .	23
3.2.1	Preprocessing . . . . .	23
3.2.2	Processing . . . . .	26
3.2.3	Post-processing and analysis of the results . . . . .	26
<b>4</b>	<b>Prismatic water tank</b>	<b>28</b>
4.1	Purpose of the analysis . . . . .	28
4.2	Analysis . . . . .	28
4.2.1	Preprocessing . . . . .	28
4.2.2	Processing . . . . .	31
4.2.3	Post-processing and analysis of the results. . . . .	32

# 1 Thin plate under dead weight



## 1.1 Purpose of the example

The goal of this project is to model how a plate deforms under its own weight and to study the rate of convergence of different type of finite elements. For that purpose, a sought solution of the stress and the displacement in the Y direction are given for some specific points. Then, in order to obtain the convergence rate of a certain type of element, the problem is solved using several meshes with different levels of refinement and the error for each of them is computed using the sought solution. The solution can then be plotted against the number of degrees of freedom of the model and a comparison between several element types can be conducted.

## 1.2 Analysis

### 1.2.1 Preprocessing

#### Geometry

The first step to model the problem is to define the geometry of the structure using the preprocessor of GiD using the geometric data given.

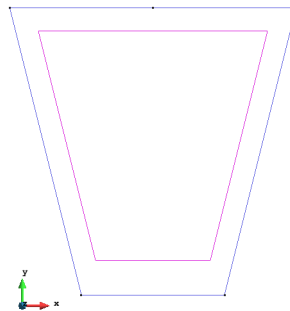


Figure 1.1: Geometry definition for Exercise 1

## Data

### Problem type

After that, the problem type is selected, in this case 'plane state' of the Ramseries Educational 2D module. In order to properly define the problem, several steps must be followed.

### Boundary Conditions

Appropriate boundary conditions need to be defined in order for the problem to be solvable. For this problem, the body is hanging from its top part (segment AC). That translates into a fixed displacement equal to 0 in both X and Y direction for the top side of the plate (segment AC).

There are no external loads applied to the structure.

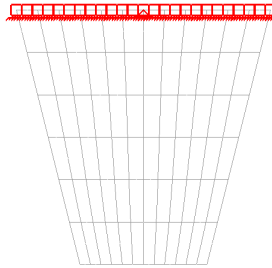


Figure 1.2: Boundary conditions for Exercise 1

### Material

It is also necessary to specify the material the body is made of and its corresponding properties. In this problem the material data is given and the only thing to do is to enter the value of the different parameters in the model, as seen in Figure 1.3

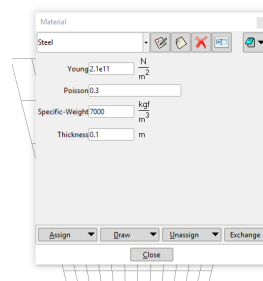


Figure 1.3: Material properties for Exercise 1

### Problem Data

In this kind of problem type, some data necessary for the analysis has to be set up. In this case, as it can be seen in Figure 1.4, the problem title is named as 'Exercise 1', no ASCII output is asked, the specific problem type selected is the plane stress model and the model is



told to consider the self weight of the body with a scale factor of 1.

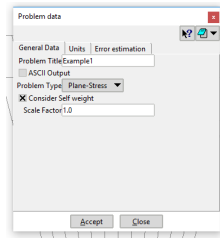


Figure 1.4: Problem Data for Exercise 1

## Mesh

To generate the several meshes used in this exercise the following options were used:

- Structured: a structured mesh is generated as GiD is able to generate structured quad and triangle meshes for this type of geometry. The number of elements assigned varies depending on the refinement desired.
- Element type: triangular or quadrilateral, depending on the mesh.
- Element order: linear elements with 3 or 4 nodes (Normal) and quadratic elements with 6, 8 and 9 nodes (Quadratic and Quadratic9) were used.

### 1.2.2 Processing

In this document, the convergence was studied for 3 node triangles, 6 node triangles, 4 node quadrilaterals, 8 node quadrilaterals and 9 node quadrilaterals. For each type of element, the problem was solved using 4 different meshes, starting with coarse meshes and proceeding to refine them.

In total, 20 different meshes were generated and then solved using the 'Calculate' option in the top menu.

### 1.2.3 Postprocessing

#### Results

The following figures show the results obtained for the different meshes in this example.

# TRIANGULAR ELEMENTS WITH 3 NODES

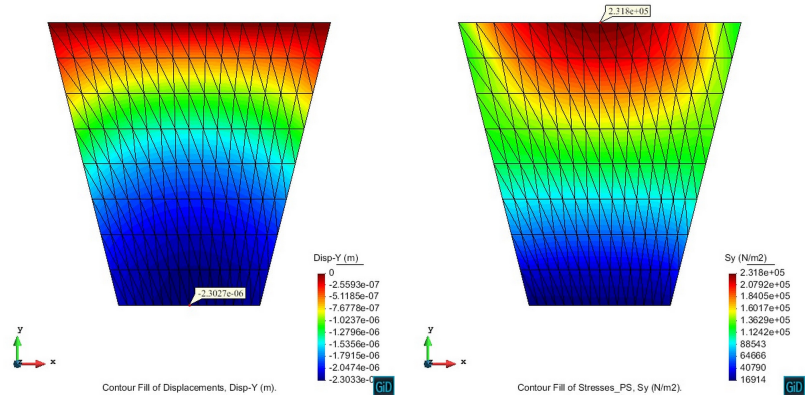


Figure 1.5: Displacement and stress for a 153-node mesh

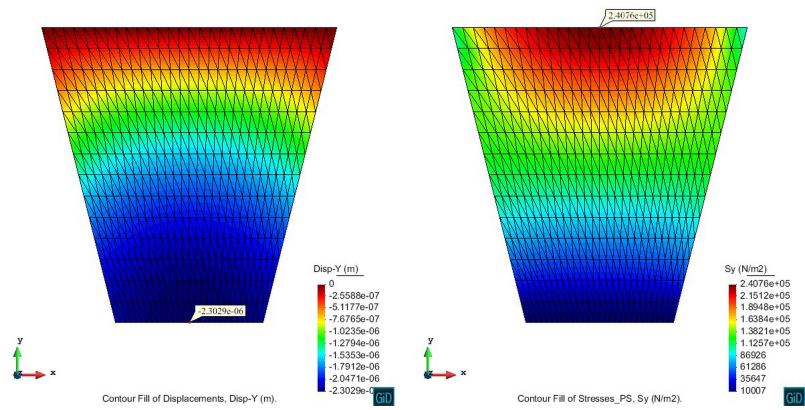


Figure 1.6: Displacement and stress for a 435-node mesh

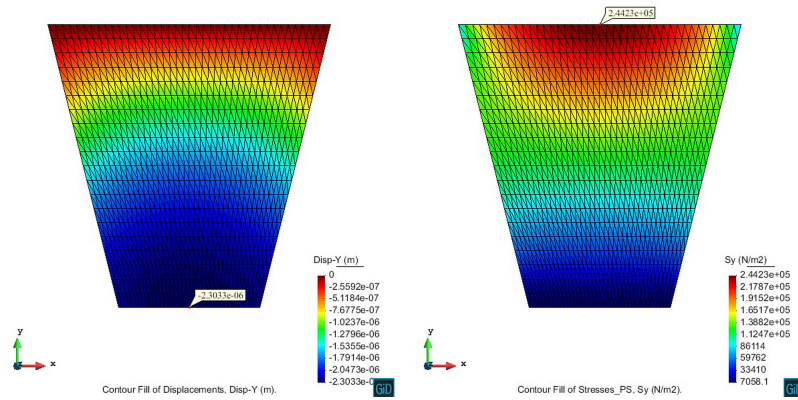


Figure 1.7: Displacement and stress for a 861-node mesh

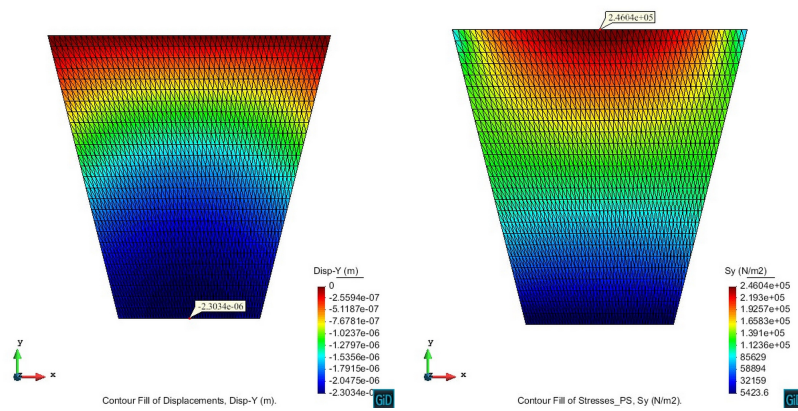


Figure 1.8: Displacement and stress for a 1431-node mesh

### TRIANGULAR ELEMENTS WITH 6 NODES

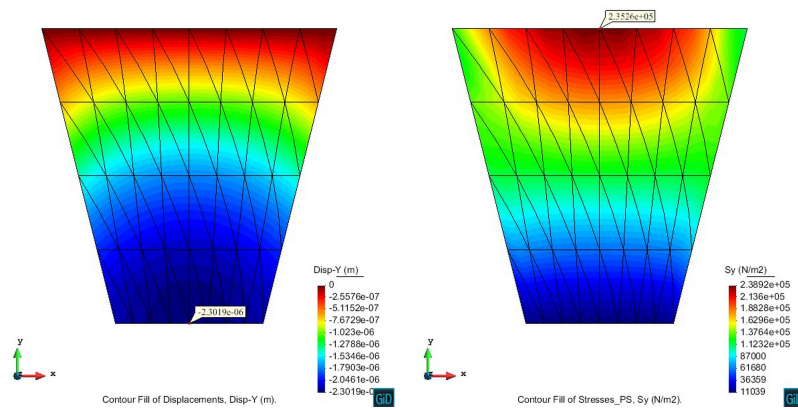


Figure 1.9: Displacement and stress for a 153-node mesh

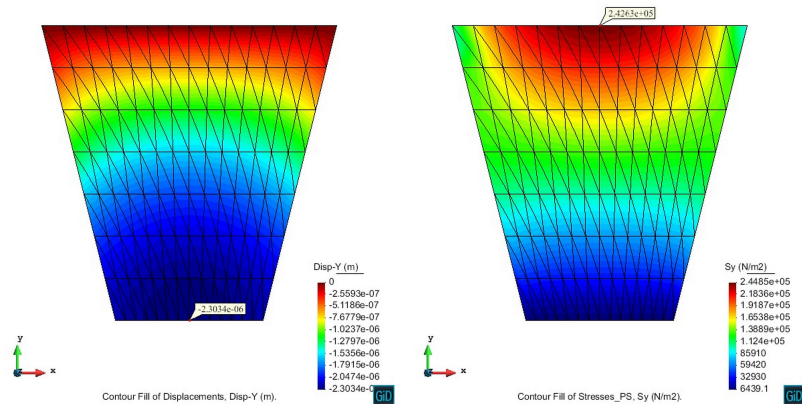


Figure 1.10: Displacement and stress for a 435-node mesh

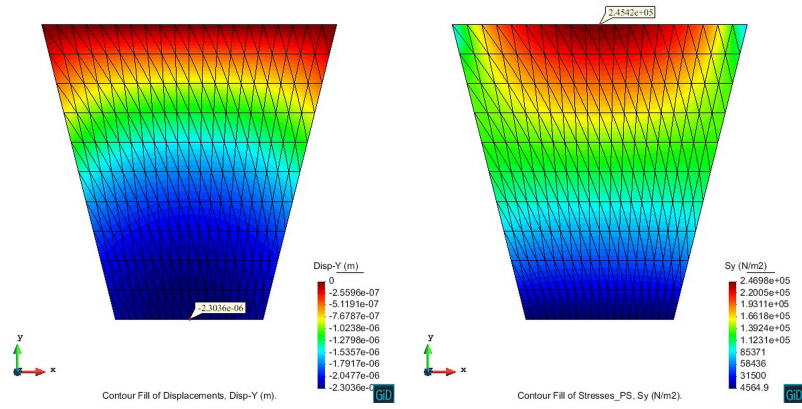


Figure 1.11: Displacement and stress for a 861-node mesh

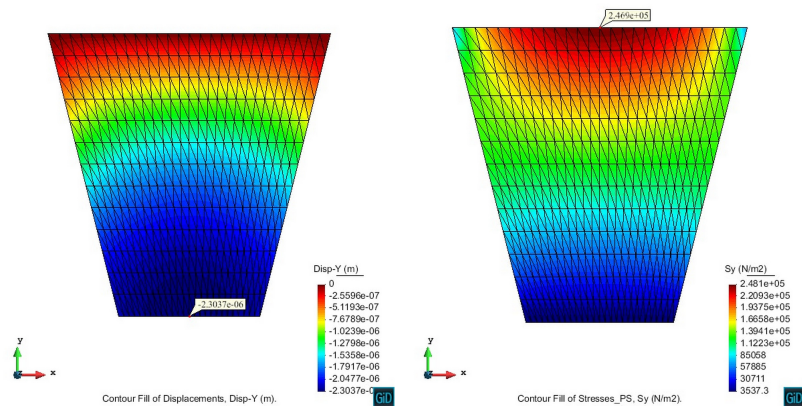


Figure 1.12: Displacement and stress for a 1431-node mesh

## QUADRILATERAL ELEMENTS WITH 4 NODES

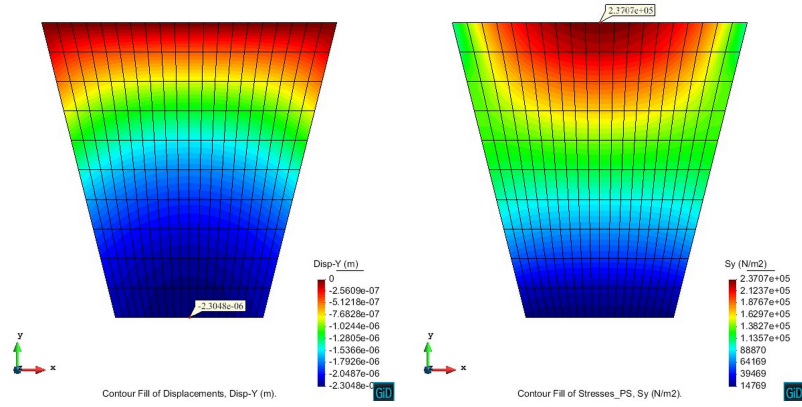


Figure 1.13: Displacement and stress for a 231-node mesh

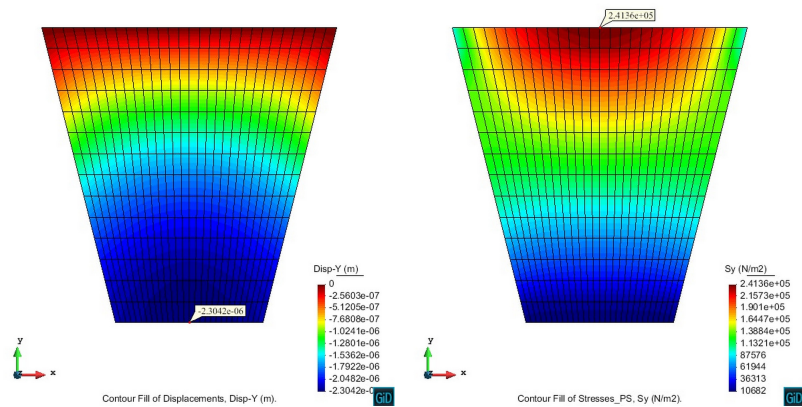


Figure 1.14: Displacement and stress for a 435-node mesh

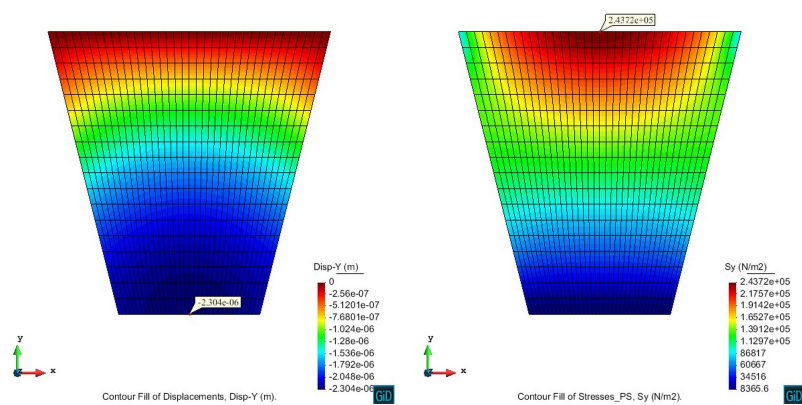


Figure 1.15: Displacement and stress for a 703-node mesh



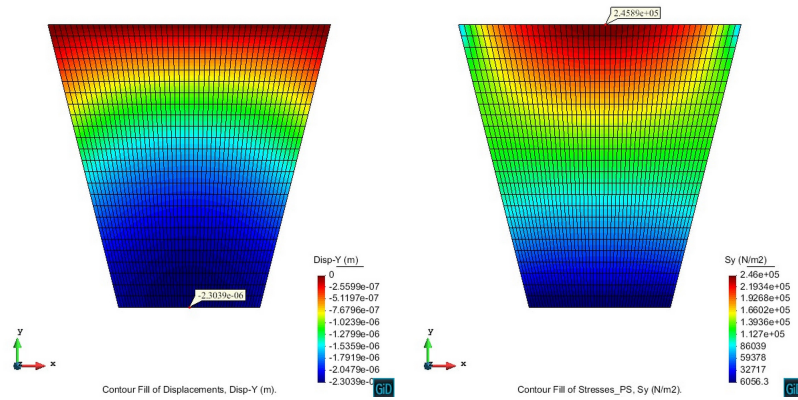


Figure 1.16: Displacement and stress for a 1326-node mesh

### QUADRILATERAL ELEMENTS WITH 8 NODES

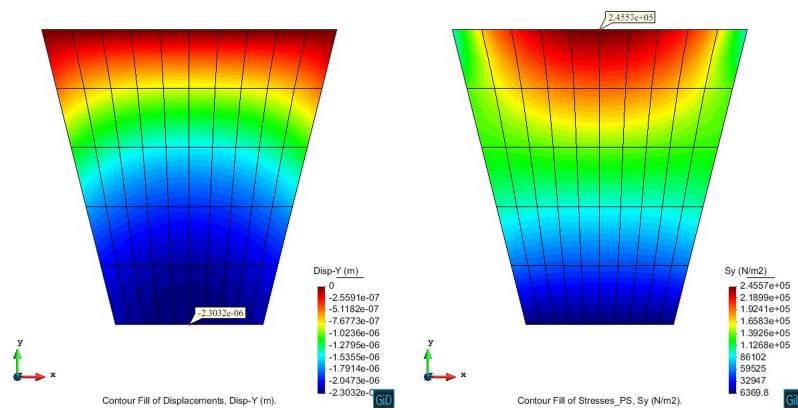


Figure 1.17: Displacement and stress for a 181-node mesh

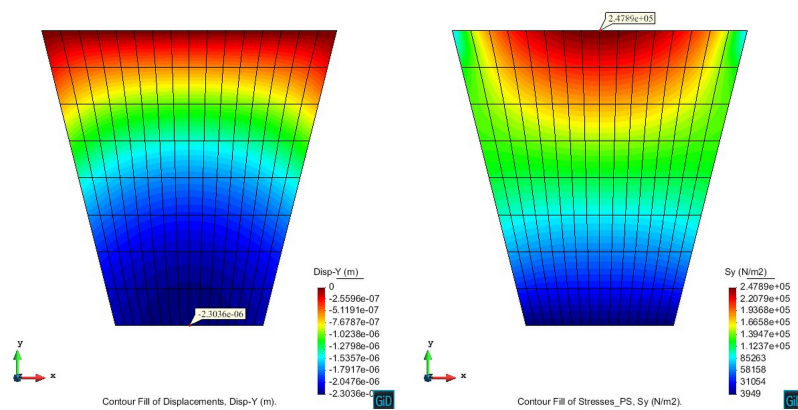


Figure 1.18: Displacement and stress for a 433-node mesh

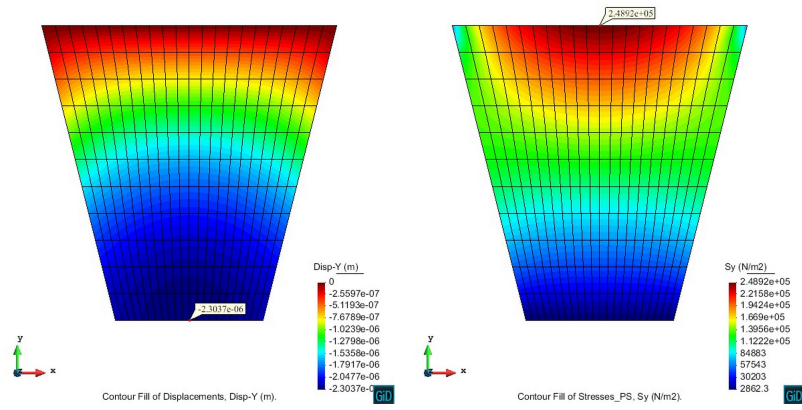


Figure 1.19: Displacement and stress for a 793-node mesh

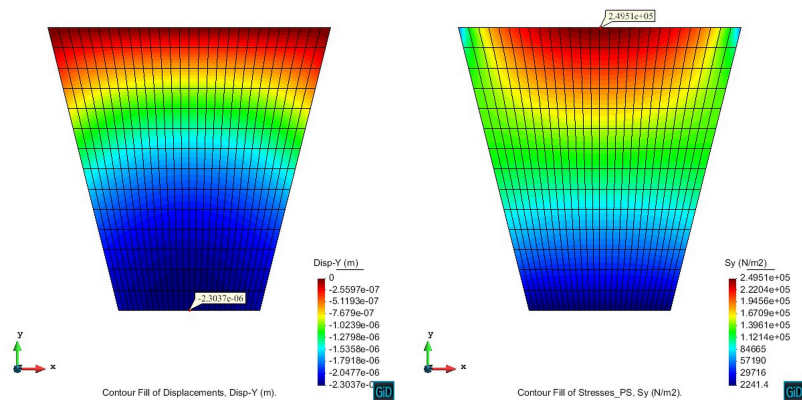


Figure 1.20: Displacement and stress for a 1261-node mesh

### QUADRILATERAL ELEMENTS WITH 9 NODES

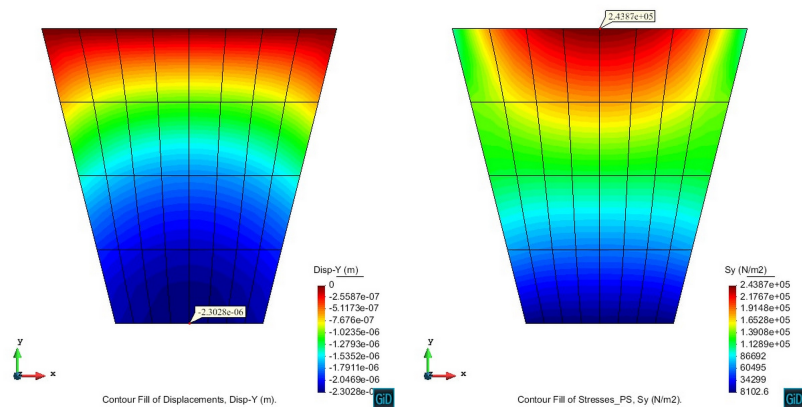


Figure 1.21: Displacement and stress for a 153-node mesh

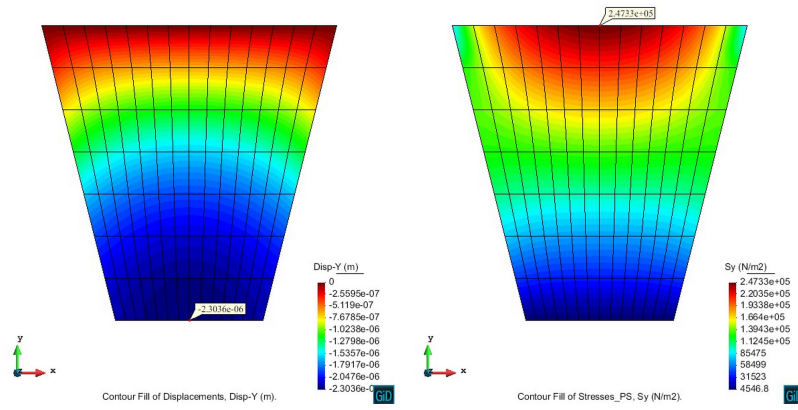


Figure 1.22: Displacement and stress for a 435-node mesh

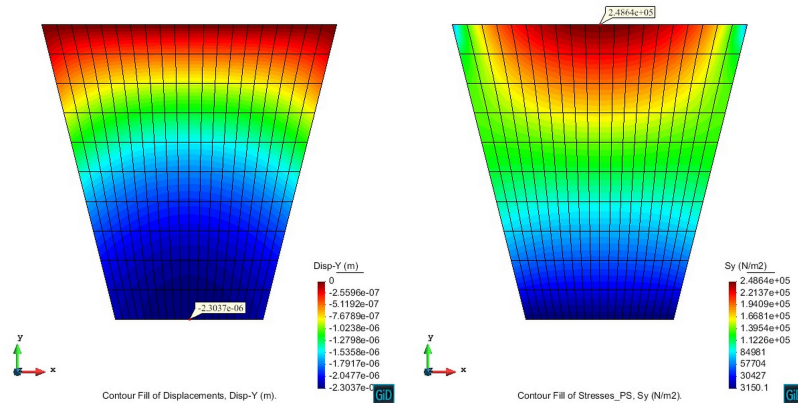


Figure 1.23: Displacement and stress for a 861-node mesh

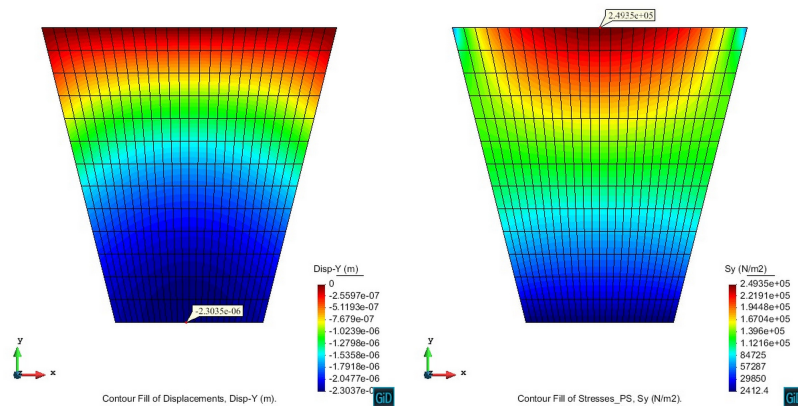


Figure 1.24: Displacement and stress for a 1431-node mesh



## Comparison of results

The table shown next presents a summary of the information obtained for all the cases that were run. The degrees of freedom of each mesh is given by GiD. The errors are computed as the relative difference between the obtained result and the sought solution and they are expressed as a percentage.

Element type	DoF	$\sigma_Y$ in B	$\epsilon_y$ in F	Error $\sigma_Y$	Error $\epsilon_Y$
			[N/m2]	[m]	
3 node triangle	153	2,310E+05	-2,3027E-06	-6,48%	1,89%
3 node triangle	435	2,408E+05	-2,3029E-06	-2,53%	1,90%
3 node triangle	861	2,442E+05	-2,3033E-06	-1,12%	1,92%
3 node triangle	1431	2,461E+05	-2,3034E-06	-0,38%	1,92%
6 node triangle	153	2,353E+05	-2,3019E-06	-4,75%	1,85%
6 node triangle	435	2,426E+05	-2,3034E-06	-1,77%	1,92%
6 node triangle	861	2,454E+05	-2,3036E-06	-0,64%	1,93%
6 node triangle	1431	2,469E+05	-2,3037E-06	-0,04%	1,93%
4 node quad	231	2,370E+05	-2,3048E-06	-4,04%	1,98%
4 node quad	433	2,414E+05	-2,3042E-06	-2,28%	1,96%
4 node quad	703	2,437E+05	-2,3040E-06	-1,33%	1,95%
4 node quad	1326	2,459E+05	-2,3039E-06	-0,45%	1,94%
8 node quad	181	2,456E+05	-2,3032E-06	-0,58%	1,91%
8 node quad	433	2,479E+05	-2,3036E-06	0,36%	1,93%
8 node quad	793	2,489E+05	-2,3037E-06	0,78%	1,93%
8 node quad	1261	2,495E+05	-2,3037E-06	1,01%	1,93%
9 node quad	153	2,439E+05	-2,3028E-06	-1,27%	1,89%
9 node quad	435	2,473E+05	-2,3036E-06	0,13%	1,93%
9 node quad	861	2,486E+05	-2,3037E-06	0,66%	1,93%
9 node quad	1431	2,494E+05	-2,3035E-06	0,95%	1,92%

Table 1.1: Summary of results for all the analyzed meshes

Figures 1.25 and 1.26 present diagrams of the convergence speed of each type of element for the displacement in point F and the stress in point B.

### 1.2.4 Analysis of results

Observing the plots of convergence presented before it can be seen that the different elements used behave as expected. That means that the lowest convergence speed corresponds to linear triangles with 3 nodes and the fastest convergence speed corresponds to quadrilaterals with 9 nodes.

The convergence plot for the displacement of point F (middle point between D and E) shows how the quadratic elements (Tri6, Quad8 and Quad9) rapidly converge to the same value, while for the linear elements (Tri3 and Quad4) the convergence is slower and they would require a finer mesh in order to obtain the same final result.

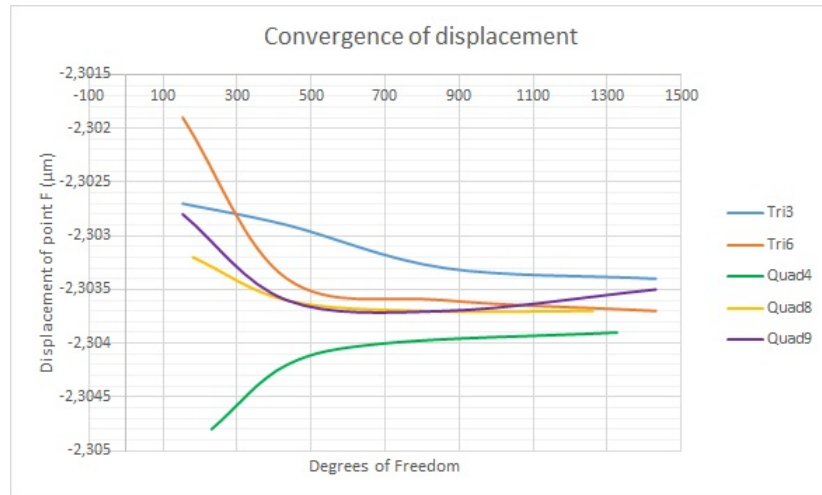


Figure 1.25: Convergence of displacement in point F for several element types

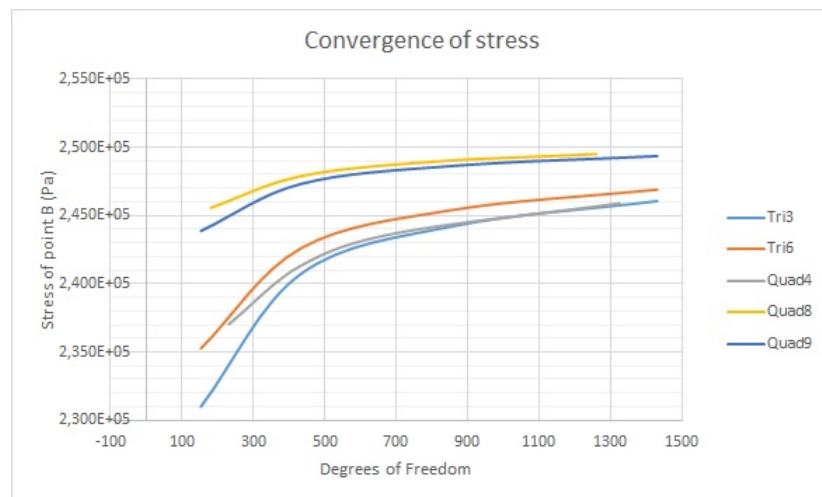


Figure 1.26: Convergence of stress in point B for several element types

This behavior can also be seen, with some slight differences, in the convergence plot for the stress of point B. In this case the best elements are the quadratic quadrilaterals, the behavior of the quadratic triangle is not as good for the stress as it was for the displacement convergence.

An important fact that should be pointed out is that both the displacement and the stress presented do not converge to the sought solution given in the exercise. Table 1.2.3 shows that the error committed is not very important, around one percent for the stress and two percent for the displacement in the worst cases.

There are several sources of error that can generate this discrepancy between the obtained and the sought solution. One possible source is that the value of gravity considered when obtaining the sought solution could be different from the one used by the Ramseries solver. It could be also possible that the sought solution was obtained using an extremely coarse mesh that was not accurate enough to converge to the actual solution.

## 2 Plate with two sections.

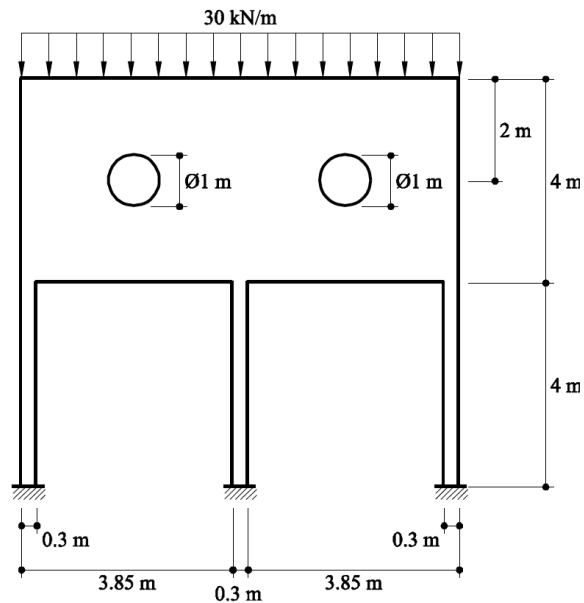


Figure 2.1: Cross section of the plate of analysis.

### 2.1 Purpose of the example

In this second exercise we aim to analyze the distribution of stresses that a drop  $\delta$  of the central column produces on a reinforced concrete plate with two holes.

### 2.2 Analysis

#### 2.2.1 Preprocessing

##### Geometry

First, we define the geometry of the structure in GiD. Please, see figure 2.2

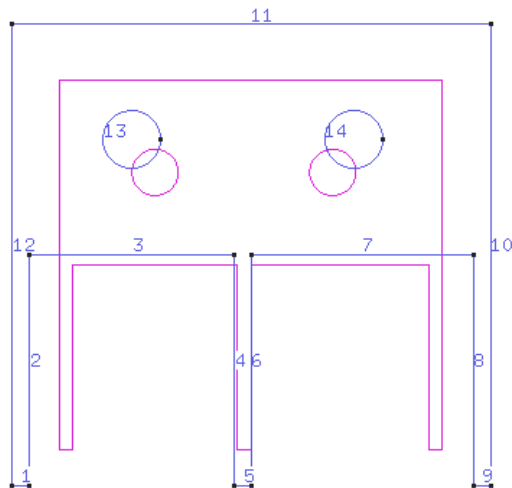


Figure 2.2: Geometry of the plate sketched using GiD. Line number reference is also included.

## Data

### Problem type

After the definition of the geometry of the structure, we can now choose the type of problem we want to solve. For this case, as we are considering a cross section of the plate, we face a plane problem (in fact it is a plane stress problem, since the plane is thin and it is loaded on its midplane. In GiD, we select,

*Data / Problem Type / RamSeries\_Eductional\_2D / Plane State*

### Boundary conditions

- Displacement constraint Linear constraint: movement in both directions  $x$  and  $y$  is prevented for the right and left columns (lines 1 and 9 respectively). For the central column (line 5) we need to perform different simulations with various prescribed displacements in the (negative)  $y$  direction.
- Loads / Line load / Uniform load: a distributed force of 30 kN/m is considered acting on line 11 of the geometry. Recall that positive value means towards the geometry.

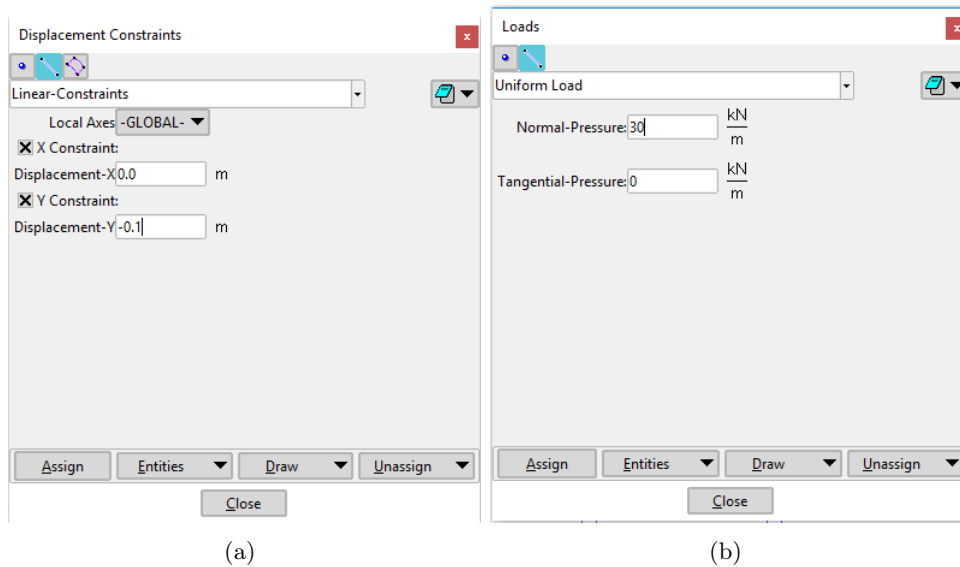


Figure 2.3: Definition of boundary conditions for the analysis of the plate.

### Material

The material considered for the plate is reinforced concrete with the following properties (see Figure 2.4 for detail):

$$E = 3.0e10 \text{ N/m}^2 \quad ; \quad \nu = 0.2 \quad ; \quad t = 0.2 \text{ m}$$

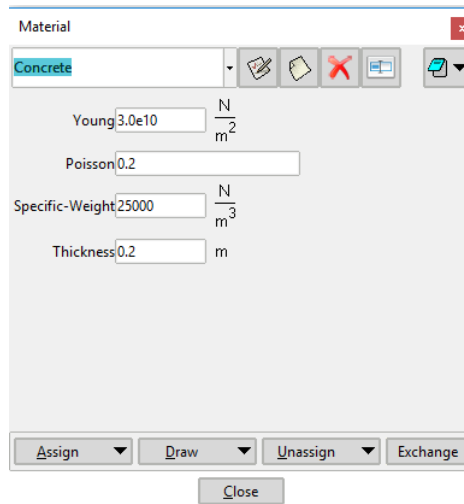


Figure 2.4: Material definition for the analysis.

### Problem Data

In this section we specify the remaining necessary data for the analysis.

- Problem Title: Exercise 2
- ASCII output: NO
- Problem type: Plane-stress
- Consider self-weight: No
- Scale factor: 1.0
- Results units: N-m-kg

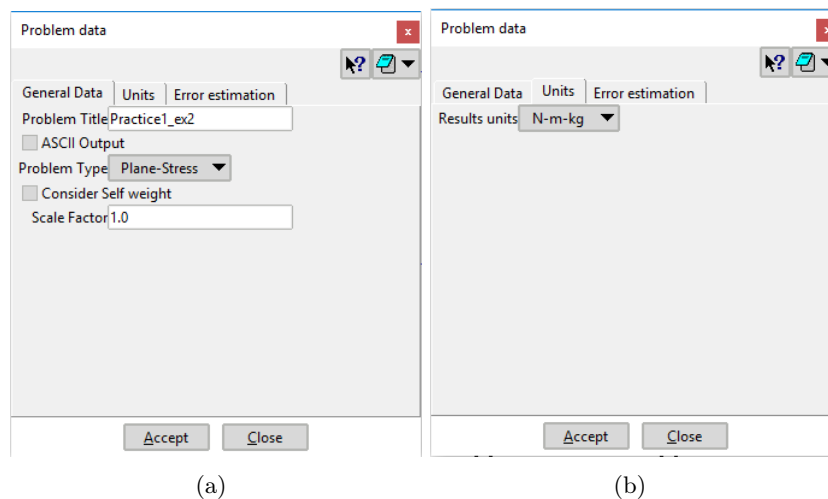


Figure 2.5: Problem data definition.

### Mesh

- Unstructured: an unstructured mesh is generated as no special requirements are set.

- Element type: triangular.
- Element order: linear elements 3 nodes.

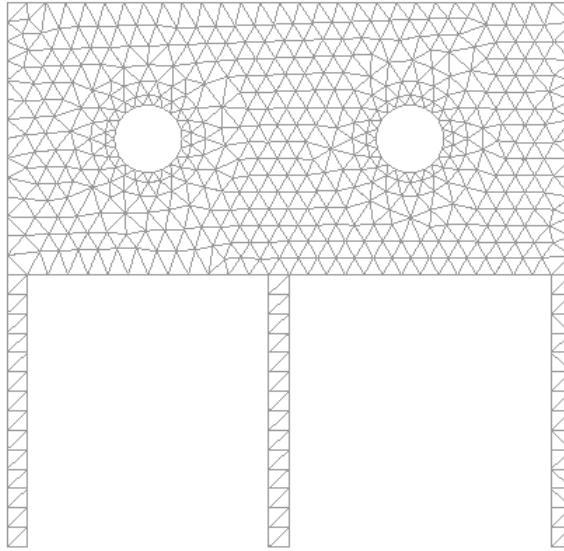
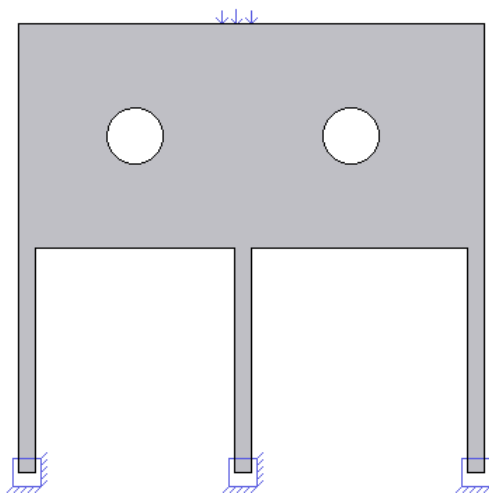


Figure 2.6: Mesh of linear triangular elements to perform the analysis of the plate.

### 2.2.2 Processing

#### Calculate

Once the mesh is successfully generated, we can proceed to run the simulation and analyze the results. Figure shows the model of our problem.



### 2.2.3 Post-processing and analysis of the results

The following figures will show the results of the analysis performed for this thin plate with three columns. As we have already said in previous section, we have considered 4 different cases for the analysis. First, we have considered the case where  $\delta_1 = 0$ , that is the case

where the three columns are completely fixed. In order to analyze the response of the central column due to the sag of the foundation, we have considered the cases:  $\delta_2 = 5$  cm,  $\delta_3 = 10$  cm and  $\delta_2 = 20$  cm.

Figures 2.7 and 2.8 are the representation of the  $\sigma_x$  and  $\sigma_y$  stresses respectively for the initial case, i.e.  $\delta = 0$ .

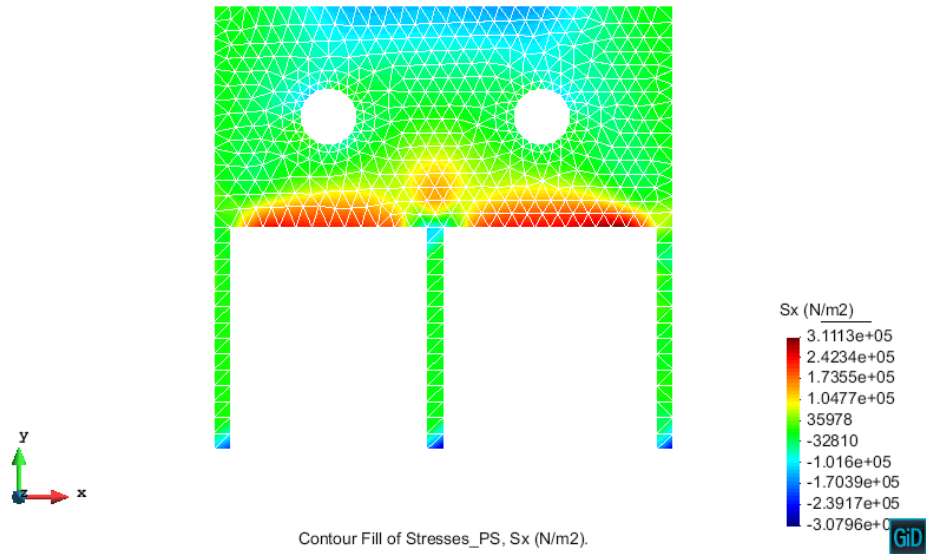


Figure 2.7: Diagram of stress  $\sigma_x$  for the initial case, when the three columns are fixed.

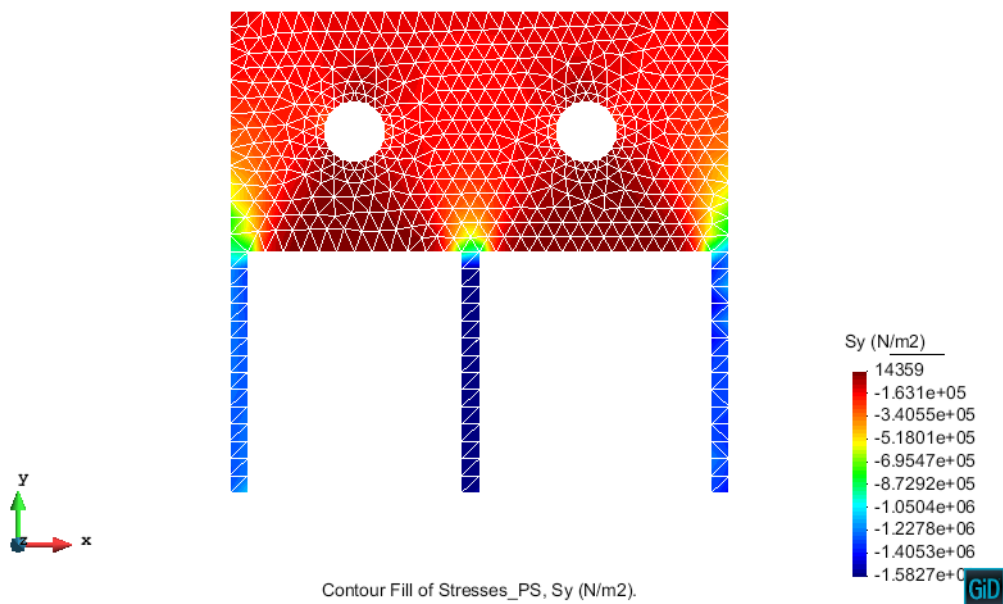


Figure 2.8: Diagram of stress  $\sigma_y$  for the initial case,  $\delta = 0$ .

As we can see, for the  $\sigma_x$  case, even though the stresses are somehow distributed over the whole structure, they concentrate on middle part of the, just where the part of the structure with the columns start. In this part, it is where there are no supports and thus there is no material to transmit the loading. On the other hand, for the  $\sigma_y$  case, the highest values of compression stresses are located in the columns, as expected since this is the part o the structure with less material and therefore, the part which can show less resistance in loading conditions.

Let's now consider the next three cases together, as they show as we can see in the figures, a similar distribution of stresses.

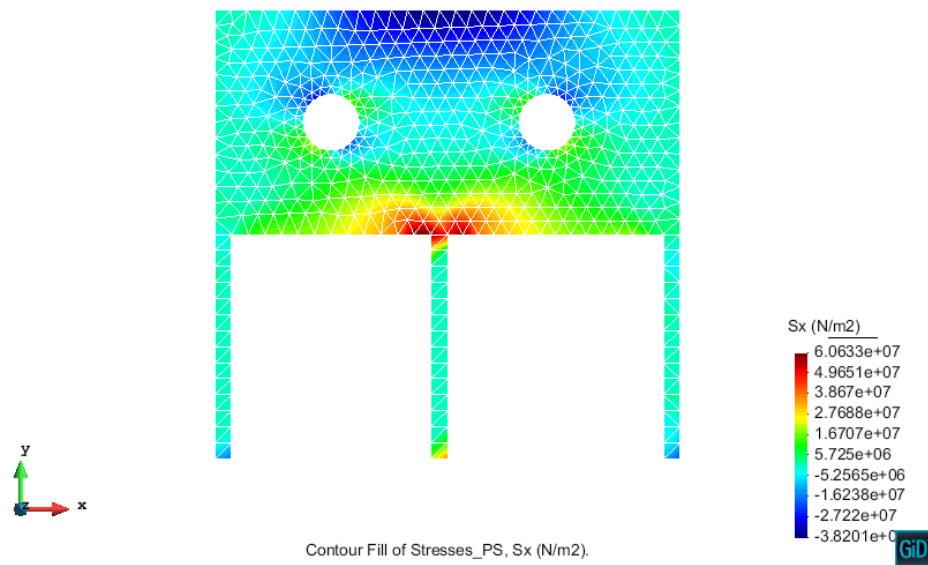


Figure 2.9: Diagram of stress  $\sigma_x$  for the case of  $\delta = 5$  cm.



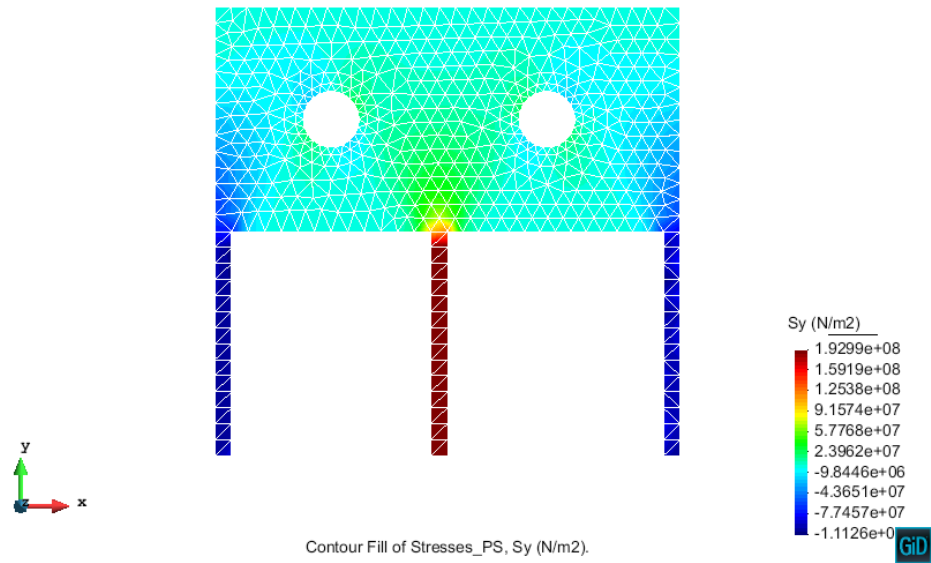


Figure 2.10: Diagram of stress  $\sigma_y$  for the case of  $\delta = 5$  cm.

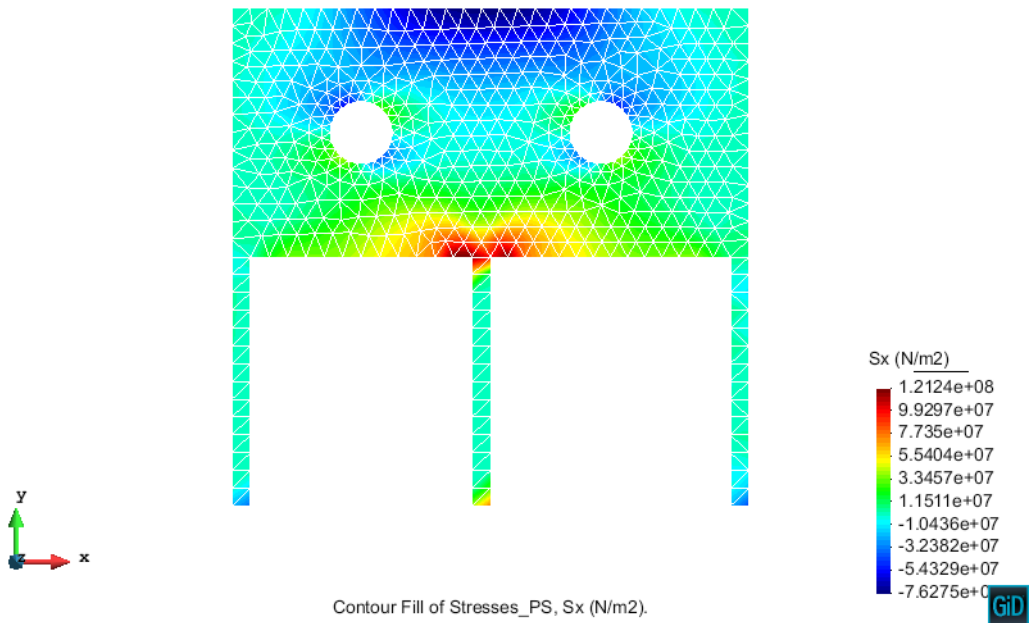


Figure 2.11: Diagram of stress  $\sigma_x$  for the case of  $\delta = 0.1$  m.

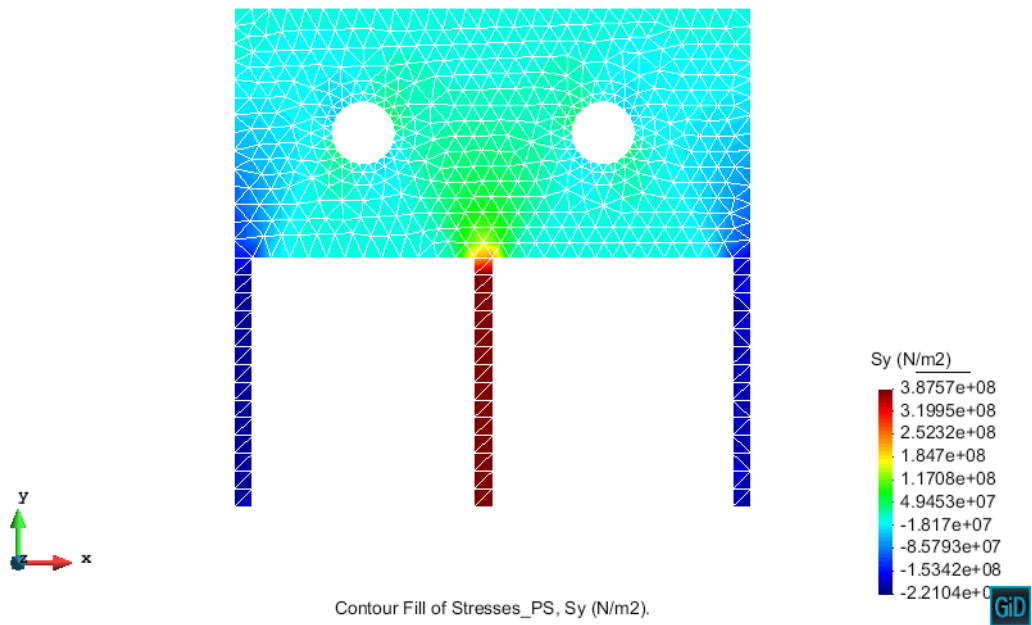


Figure 2.12: Diagram of stress  $\sigma_y$  for the case of  $\delta = 0.1$  m.

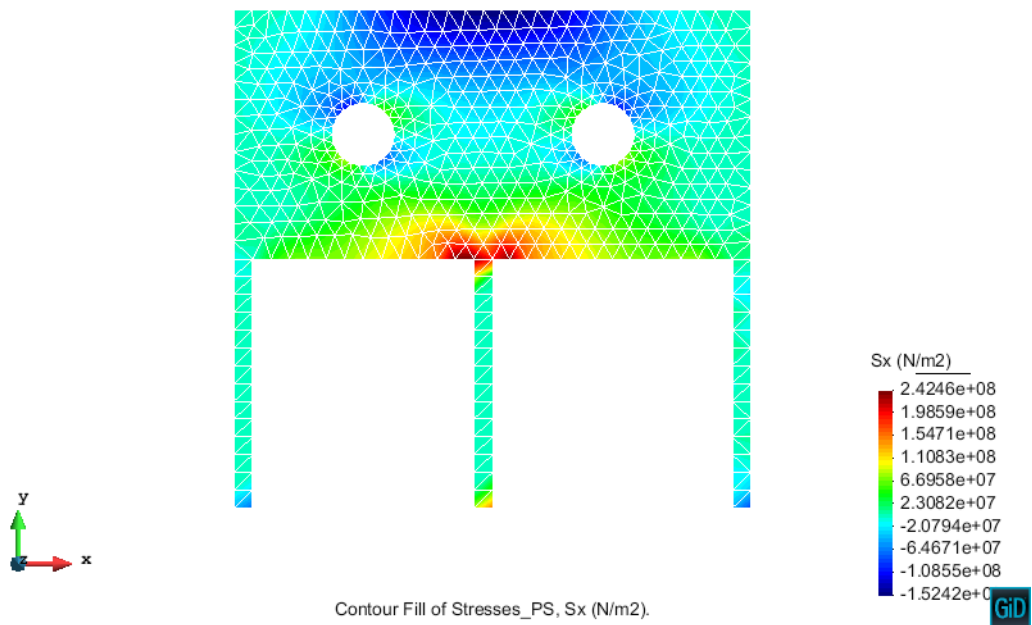


Figure 2.13: Diagram of stress  $\sigma_x$  for the case of  $\delta = 0.2$  m.

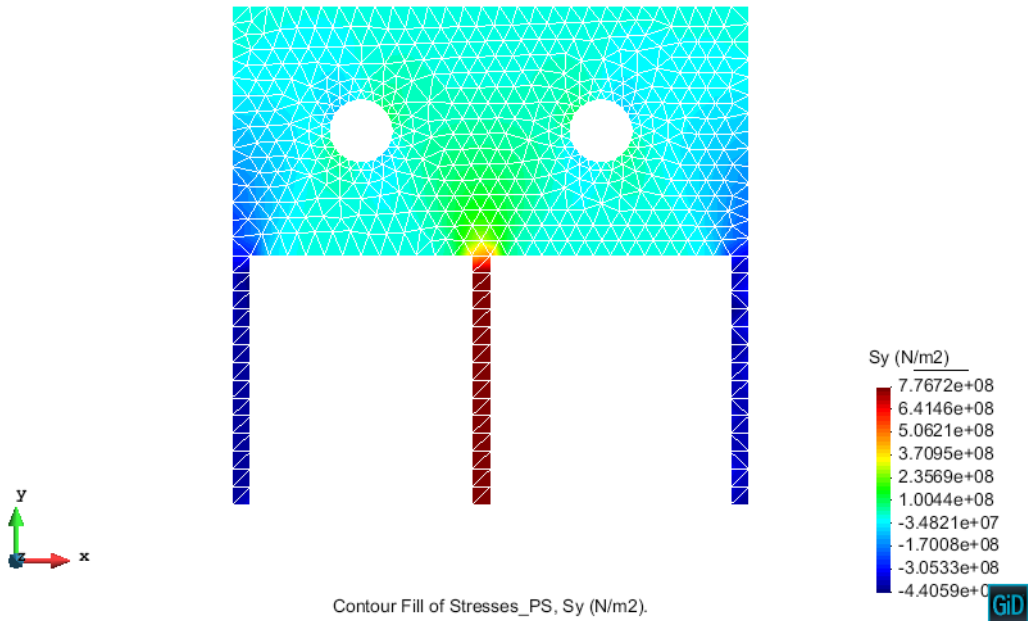


Figure 2.14: Diagram of stress  $\sigma_y$  for the case of  $\delta = 0.1$  m.

The distribution of stresses for both  $\sigma_x$  and  $\sigma_y$  is pretty much the same for the three cases under consideration. Obviously, quantitatively speaking, the results are not the same as a bigger displacement of the central column derives into a bigger level of stress.

In the case of  $\sigma_x$ , stresses concentrate in the part of the structure where the central column is placed. This is reasonable because, as the central column starts to go down, this part of the structure is the first one affected due to the sag and bigger traction stresses are concentrated here. As we see, maximum value for traction is about 242 MPa for the case of  $\delta = 0.2$ m.

For the case of  $\sigma_y$ , we obviously expect that the central column undergoes a big value of traction, since the sag pulls down the column. This fact is remarkable, but also the fact that biggest compression are located on the other two columns. This makes sense because when the central column undergoes a vertical displacement, the other two columns have to support the remaining part of the structure. Obviously, once the limit stresses for concrete are reached in the columns, the structure will collapse. In fact, we expect the collapse to happen in the central column due to the bigger resistance of concrete under compression compared to traction.

Finally, it is noticeable that the distribution of stresses is completely symmetric. This is something that we could expect a priori, since the structure is completely symmetric and it is loaded under symmetric conditions.

### 3 Plate with ventilation hole

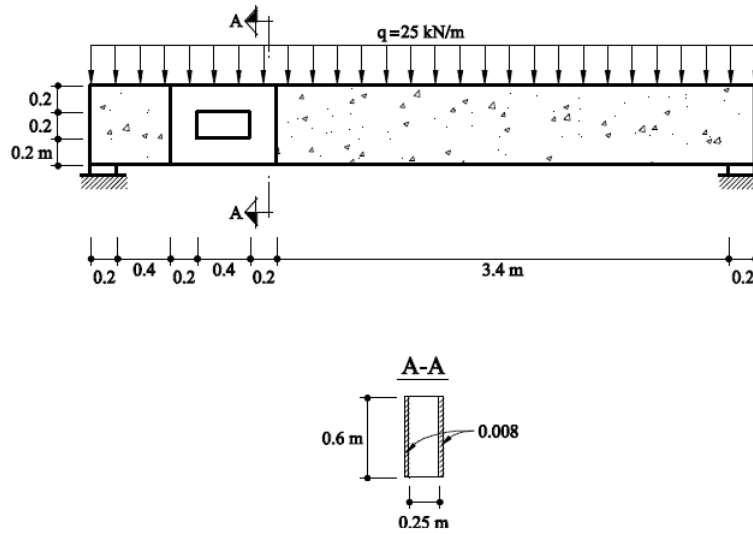


Figure 3.1: Scheme and cross section of the plate to be analyzed

#### 3.1 Purpose of the example

The purpose of this exercise is to study the addition of the metal reinforcement sheet and to check if it actually takes more stress than the concrete structure, which is the objective of the modification in the first place.

#### 3.2 Analysis

##### 3.2.1 Preprocessing

###### Geometry

The first step is to define the geometry in the preprocessor. In this case, as the structure is made of two different materials, two separated layers are created so they can have different materials assigned to each other.

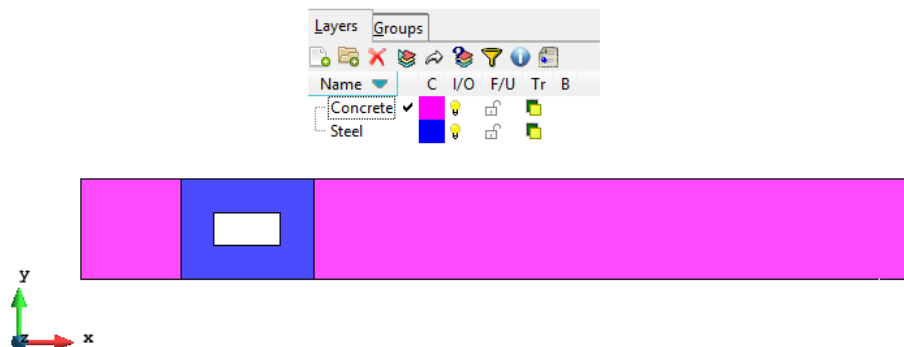


Figure 3.2: Creation of separate layers for each structure part

## Data

### Problem type

Once the geometry of the problem is created, the type of problem to be solved must be chosen. As this is a 2D simplification of reality, the case chosen is the 'plane state' of the RamSeries Educational 2D module.

*Data / Problem Type / RamSeries\_Eductional\_2D / Plane State*

### Boundary conditions

In this problem there are two different type of boundary conditions to be assigned: displacement constraints and applied loads.

- Displacement constraint / Linear constraint: in order to accurately represent reality, the segments representing the simple supports are assigned a restriction in displacement: no movement in any direction (fixed displacement = 0).

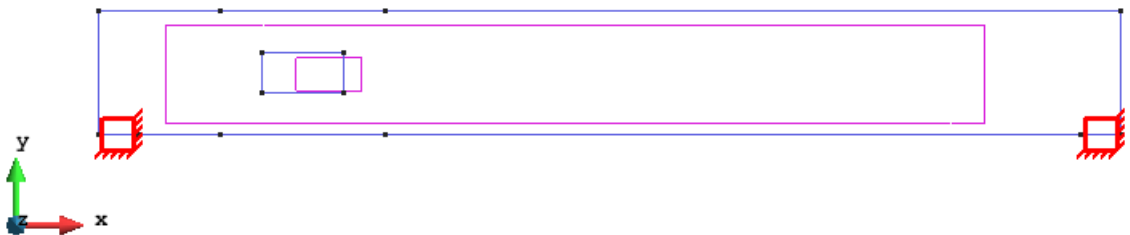


Figure 3.3: Displacement restrictions of exercise 3

- Load / Line loads / Uniform load: then a uniform load of 25 kN/m is assigned to the top segments of the structure, as seen in the next figure.

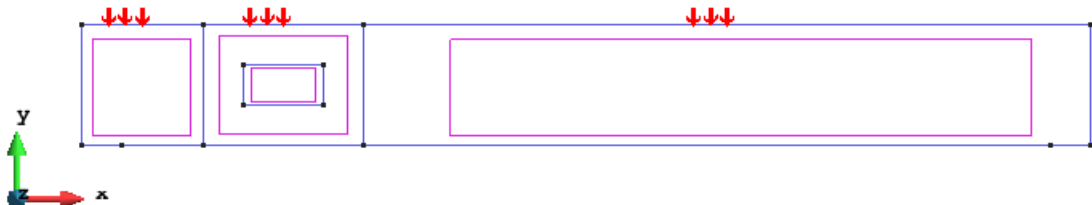


Figure 3.4: Applied external loads of exercise 3

## Material

This structure requires that two different type of materials are defined. Each material has a different thickness assigned, corresponding to the given geometrical data.

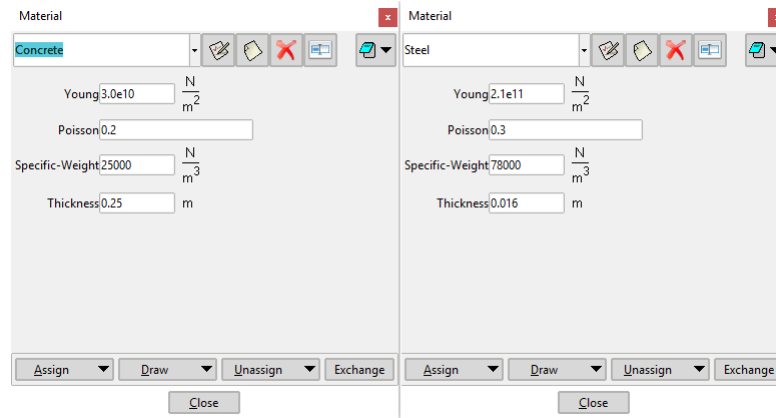


Figure 3.5: Applied external loads of exercise 3

Then each of the layers previously defined is assigned with the appropriate material, as the next figure shows.

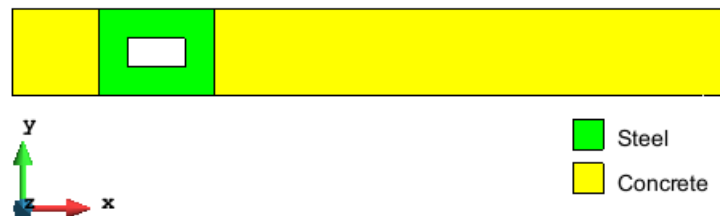


Figure 3.6: Applied external loads of exercise 3

## Problem Data

The problem data is the section where some specific data necessary for the analysis has to be defined. In this case the problem title is named as 'Exercise 3', no ASCII output is asked, the specific problem type selected is the plane stress model and the model is told to ignore the self weight of the body with a scale factor of 1.

## Mesh

To generate the mesh used in this exercise the following options were used:

- Unstructured: an unstructured mesh is generated assigning an element size of 0.2 m.
- Element type: quadrilateral.

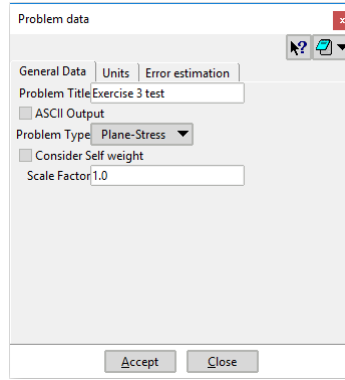


Figure 3.7: Applied external loads of exercise 3

- Element order: linear elements 4 nodes (Normal).

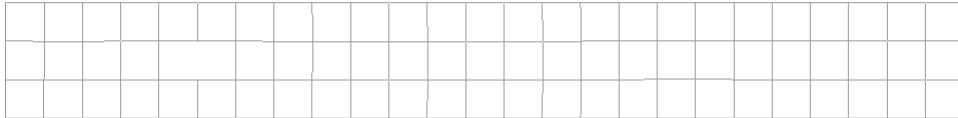


Figure 3.8: Mesh generated for exercise 3

### 3.2.2 Processing

Once everything is defined, the problem can be calculated and the results obtained.

### 3.2.3 Post-processing and analysis of the results

The following figures show the results obtained for this exercise.

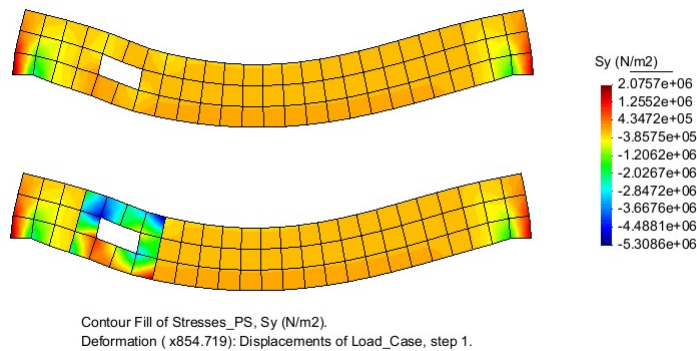


Figure 3.9: Stress distribution plotted on deformed structure

As it can be seen in 3.9, the metal reinforcement is sustaining a higher stress than the concrete structure. That was exactly the point of adding the reinforcement: get the metal

sheet to take part of the stresses that the structure would have to take and that would probably make it fail.

It is interesting to notice that for the concrete structure the highest stresses are located where the simple supports are, and not around the hole as it would have been the case without the reinforcement.

Also, by the sign of the stresses it is possible to say that the concrete plate is working mostly under compression and the metal sheet is working under tension. That means the structure plate+reinforcement is working appropriately, as each material is working under its best capabilities and best range of behavior (compression for concrete and tension for metal).

The deformed shape is as expected for a plate simply supported at the ends with a uniform load distributed over the top.



## 4 Prismatic water tank

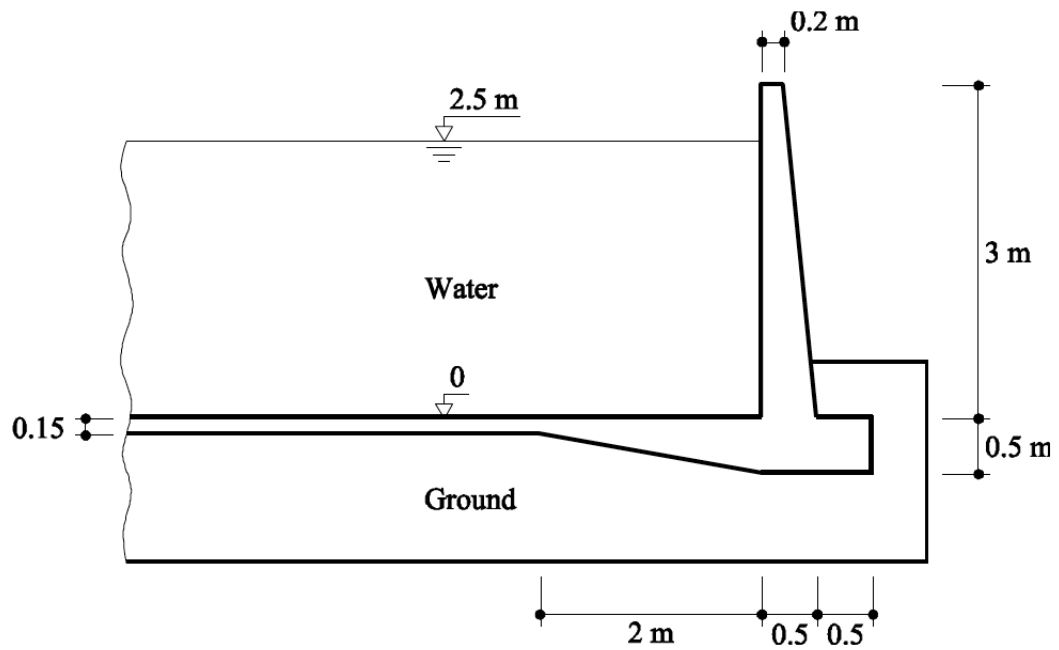


Figure 4.1: Cross section of the prismatic water tank for the analysis.

### 4.1 Purpose of the analysis

In this last exercise, we aim to analyze the state of stress of the cross section of a rectangular water tank made of reinforced concrete.

### 4.2 Analysis

#### 4.2.1 Preprocessing

##### Geometry

First, we need to define the geometry of the structure by using the GiD sketcher tool.

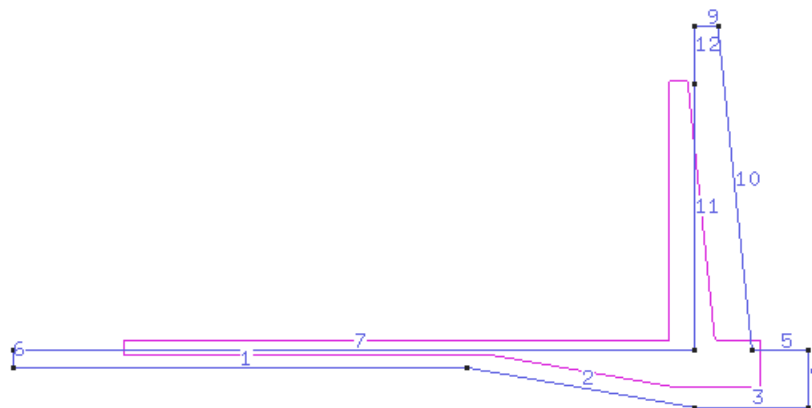


Figure 4.2: Geometry of the structure sketched using GiD.

## Data

### Problem type

After defining the geometry, we now are able to decide the type of problem we want to solve. For this case, as we are considering a cross section of the structure by means of symmetry conditions, we face a plane problem. Therefore, we choose,

*Data / Problem Type / RamSeries\_Eduational\_2D / Plane State*

### Boundary conditions

The type of boundary conditions that should be considered in this problem are the following:

- Displacement constraint / Linear constraint: symmetry boundary condition on the axis of revolution of the water tank is imposed. This condition restricts the motion in the  $x$  direction of the points laying of the symmetry axis.
- Elastic constraint / Linear - Elast. constraints: an elastic constraint is assigned to the lines 1, 2 and 3 on the geometry, i.e. the lines laying on the ground. This enables to simulate a more realistic scenario.
- Loads / Line load / Uniform load: distributed load is applied to line 7 of the geometry, due to the water weight inside the tank. The value is (unit of length):

$$p = \rho gh = 1000 \cdot 9.81 \cdot 2.5 = 24525 \text{ N/m}^2$$

or simply 24525 N/m accounting for a unit length in the transversal direction. Recall that pressure is positive when it goes towards the geometry.

- Loads / Line load / Linear load: also we need to account for the hydrostatic pressure distribution in line 11 of the geometry. We need to assign both limit values for the load and en points.

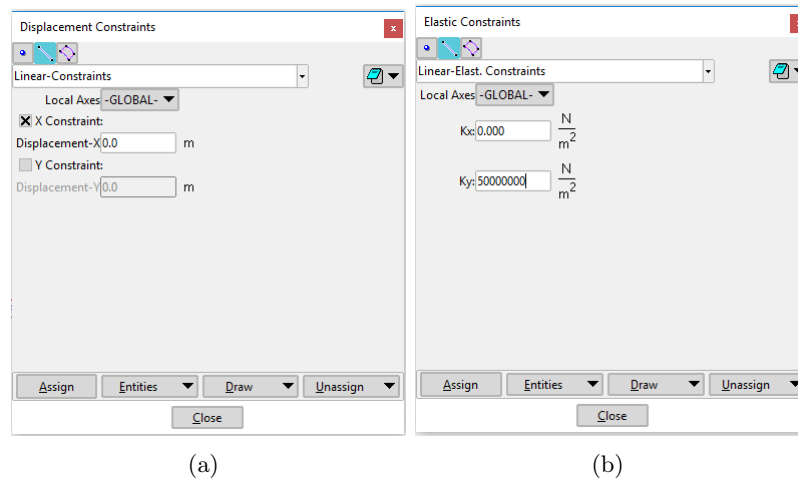


Figure 4.3: Definition of displacement constraint for the symmetry axis and the elastic constraint for the ground.

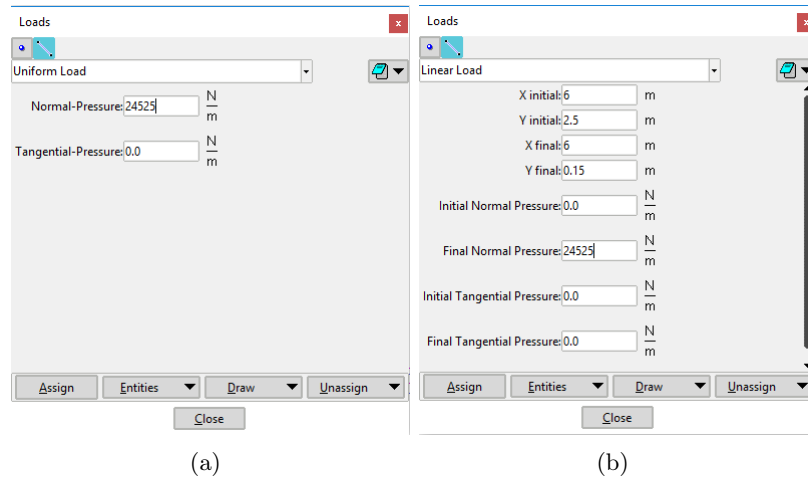


Figure 4.4: Loading definition in GiD for the analysis.

## Material

The material of the structure is reinforced concrete with the following data:

$$E = 3.0e10 \text{ N/m}^2 \quad ; \quad \nu = 0.2 \quad ; \quad \gamma = 24000 \text{ N/m}^3$$

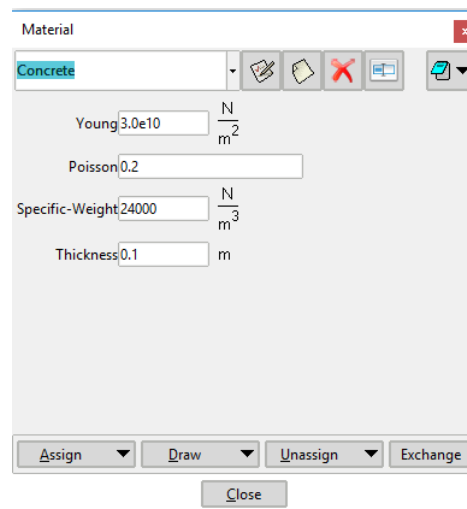


Figure 4.5: Definition of the material properties for the reinforced concrete.

## Problem Data

In this section we specify the necessary data for the analysis.

- Problem title: Exercise 4
- ASCII output: No
- Problem type: Plane-strain.
- Consider self-weight: Yes. We have decided to consider the self-weight in this case in order to more accurately represent the reality of the problem.
- Scale factor: 1.0

- Results units: N-m-kg

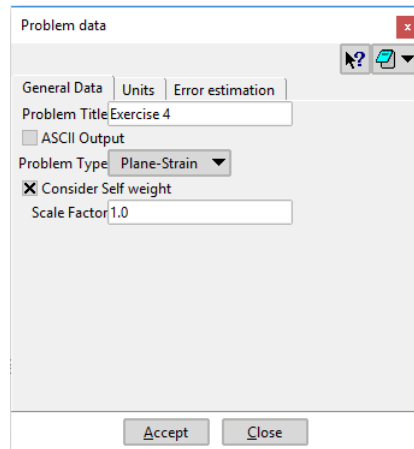


Figure 4.6: Problem data definition

## Mesh

The mesh is set with the following specifications:

- Unstructured: an unstructured mesh is generated.
- Element type: quadrilateral.
- Element order: linear elements 4 nodes (Normal).

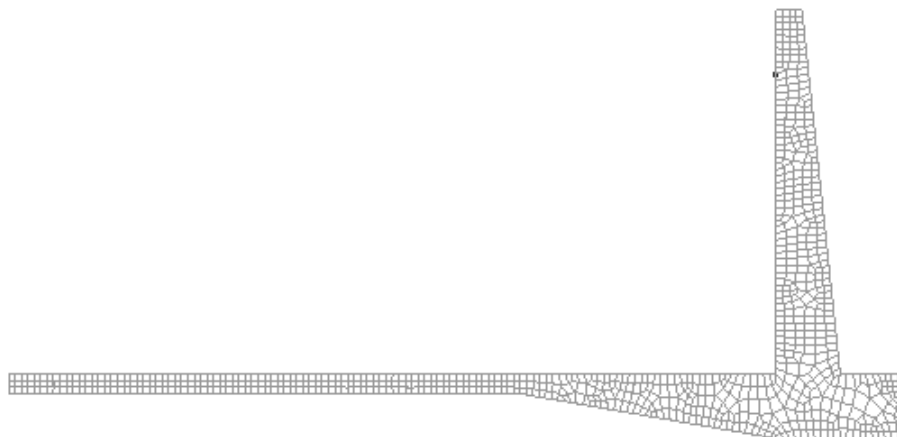


Figure 4.7: Mesh of linear quadrilateral elements for the analysis.

### 4.2.2 Processing

#### Calculate

Once the mesh is completed, we can proceed to calculate the problem and head to the post-processing part. Figure 4.8 shows the actual problem to solve.

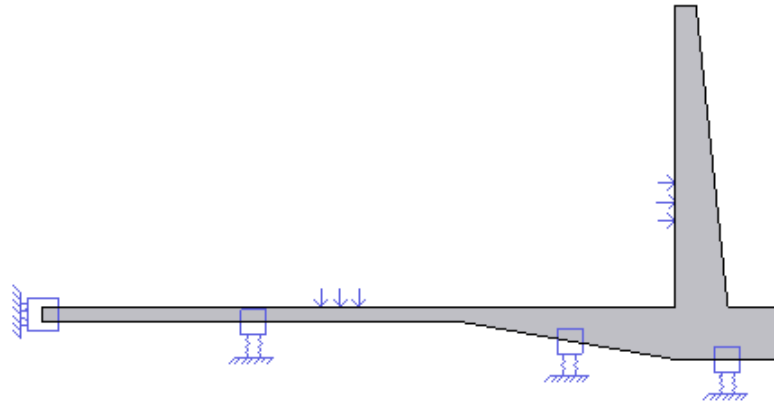


Figure 4.8: Model of the water tank.

#### 4.2.3 Post-processing and analysis of the results.

The following figures show the results of the analysis sought after in the analysis of the water tank. Even though it is not needed, Figure 4.9 showing the  $y$  component of the displacements is including, just to check that the results makes sense. As we can see, the largest vertical displacements are located in the part where there is more concrete.

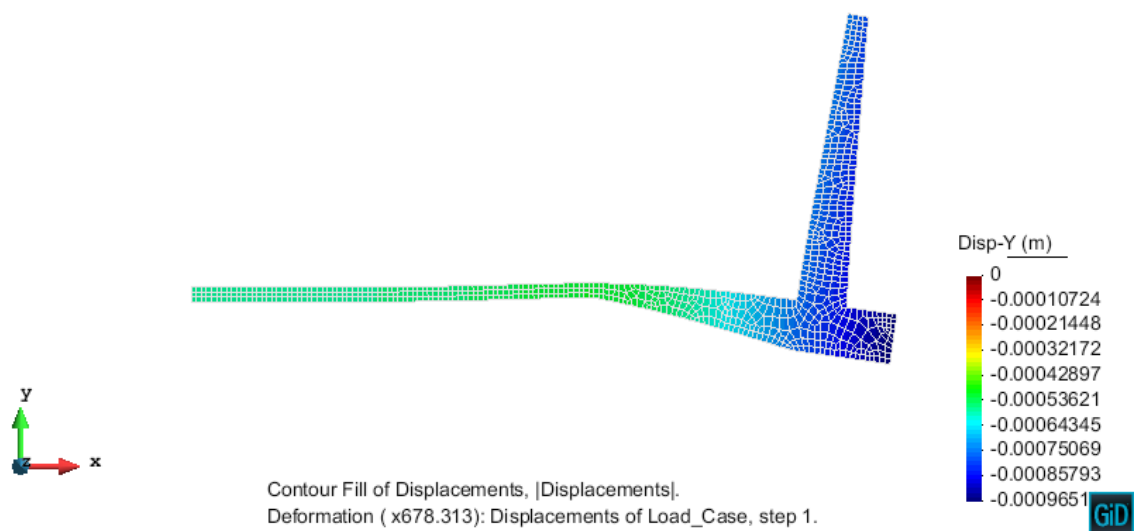


Figure 4.9: Vertical displacements on the structure,  $u_y$ .

Figures 4.10 and 4.11 show respectively the  $x$  and  $y$  component of the stresses. As we can see the highest values of stress are located within the sharp corner of the geometry, which

is a usual result. Other than that, the distribution in the  $\sigma_y$  case is somewhat uniform but concentrated on this corners. The pressure applied by the water inside will tend to curve the tank which gives an explanation about why the  $\sigma_y$  stresses concentrate on the lower-right part of the tank.

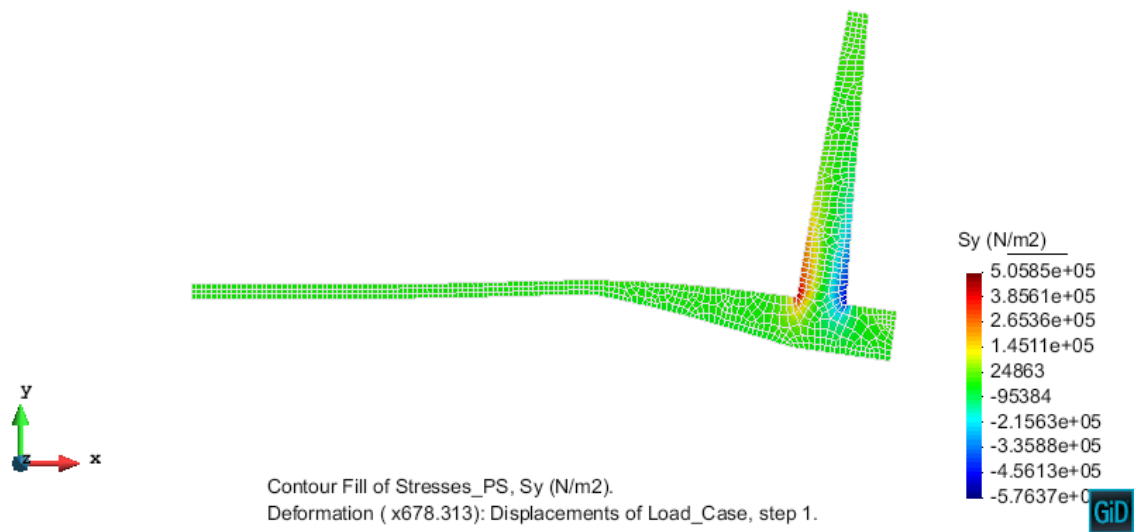


Figure 4.10: Diagram of stresses  $\sigma_y$  of the cross section.

In the case of the  $\sigma_x$  stresses, it is noteworthy the presence of large compression stresses in the lower part of the structure (blue zone) mainly due to the fact that the structure tends to get curved as a consequence of the loading.

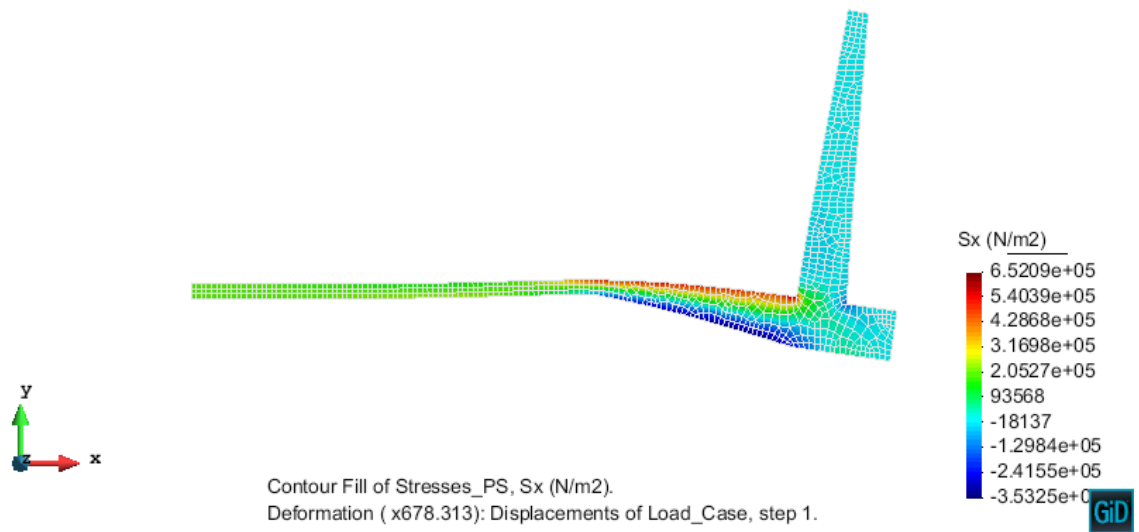


Figure 4.11: Diagram of stresses  $\sigma_x$  for the water tank.