UPC - BARCELONA TECH
MSc Computational Mechanics
Spring 2018

Computational Structural Mechanics and Dynamics

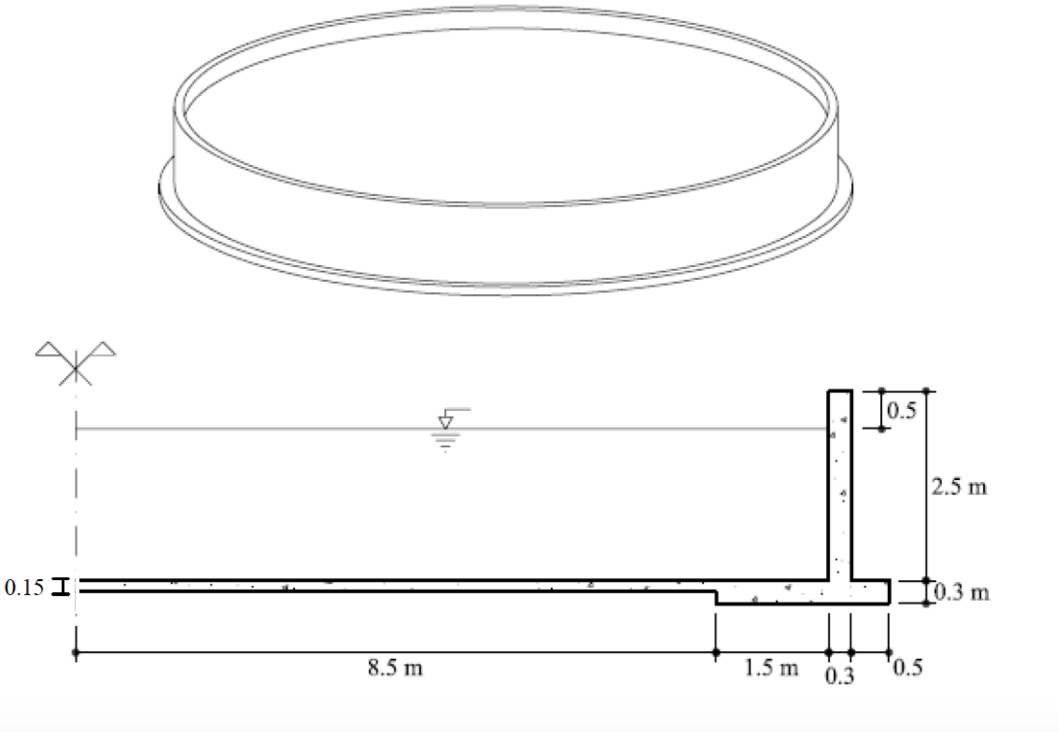
GiD Homework 2

Due 6/04/2018

Prasad ADHAV

Contents

1 Circular tank



1.1 Purpose of the example

In this first exercise we aim to analyze the structural behaviour of a concrete tank which is used to storage water in a purification plant.

1.2 Analysis

1.2.1 Preprocessing

Geometry

First, we define the geometry using the GiD sketcher tool.

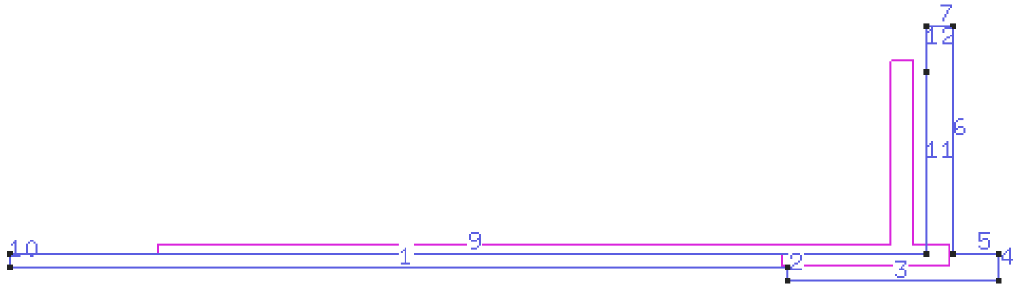


Figure 1.1: Geometry of the concrete tank.

Data

Problem type

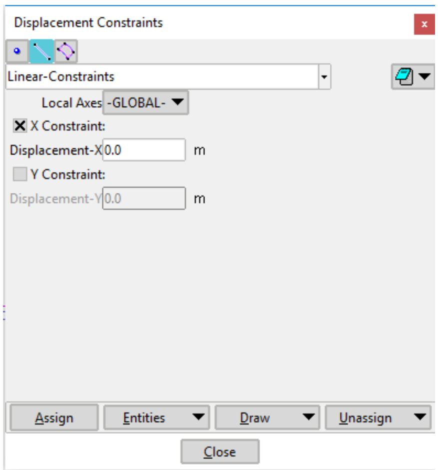
Once the geometry is defines, we can now choose the type of problem that must be solved using *RamSeries*. For this case, we are facing a axisymmetric problem since we consider a cross section of revolution. Thus, we choose

$$Data/ProblemType/Ramseries\_Educational\_2D/Rev\_Solids$$

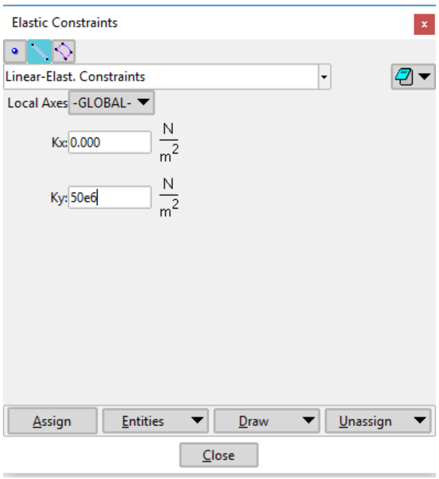
Boundary Conditions

The type of boundary conditions that are considered in this example are the following:

- Displacement Constraints / Linear-Constraints: for line 10, the one corresponding to the axis of symmetry, motion in the x direction is prevented to simulate the symmetry conditions of the structure.
- Elastic Constraints / Linear Elast.-Constraints: a ballast coefficient needs to be applied in lines 1 and 3, the ones laying directly on the ground.



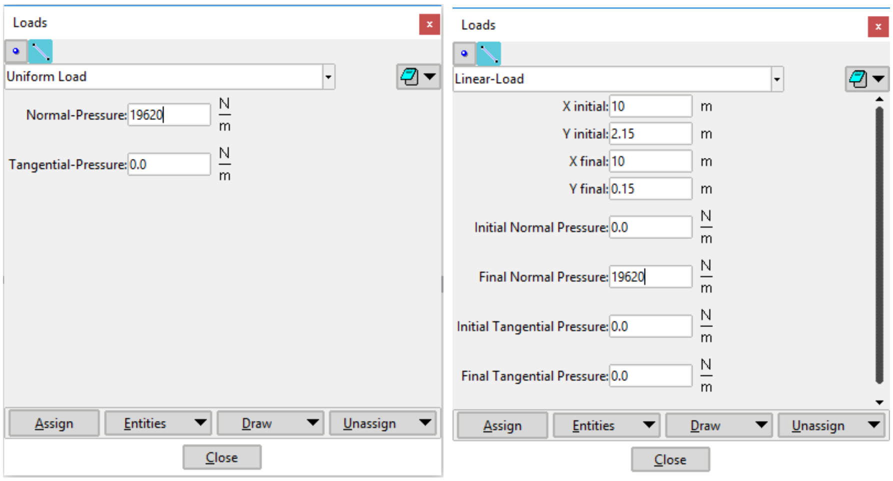
(a)



(b)

Figure 1.2: Definition of displacement and elastic constraints.

- Load / Uniform Load : a uniform distributed load of value $\rho g h = 19620$ N/m is considered acting on line 9 , to simulate the water inside the tank.
- Load / Linear Load: distributed linear load is applied with value 0 on the top of the water level and value 19620 N/m on the bottom. This is the hydrostatic pressure acting over line 11.



(a) (b)

Figure 1.3: Definition of both uniform and linear distributed loads on the tank.

Material

The tank is made of concrete with the following mechanical characteristics:

$$E = 3.0e10 \qquad N/m^2 \qquad ; \qquad \nu = 0.2$$

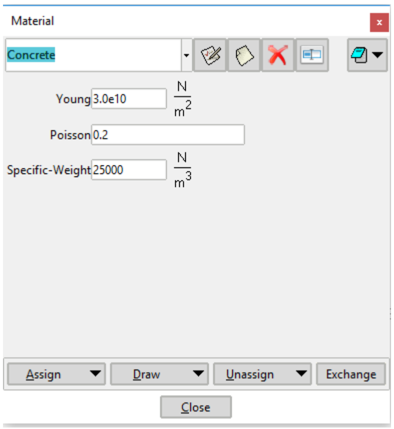


Figure 1.4: Material of the tank on GiD.

Problem Data

In this section we specify some additional data required for the analysis.

- Problem title: Exercise1
- ASCII Output: NO
- Consider self weight: Yes
- Scale factor: 1.0
- Result Units: N-m-kg

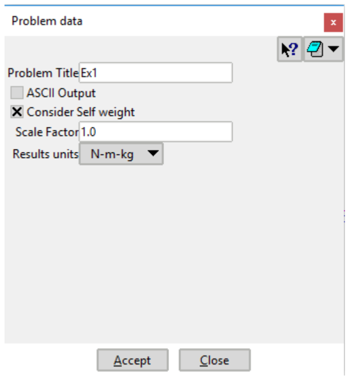


Figure 1.5: Problem data definition for exercise 1.

Mesh

We consider the following mesh for the simulation:

- Unstructured: we consider an unstructured mesh (non spetial requirements are set on the statement of the problem).
- Element type: quadrilaterals
- Linear element: quadrilaterals with 4 nodes.

Figure 1.6: Mesh of linear quadrilateral elements for the simulation of exercise 1.

1.2.2 Processing

Once the mesh is generated, we proceed to calculate the problem.

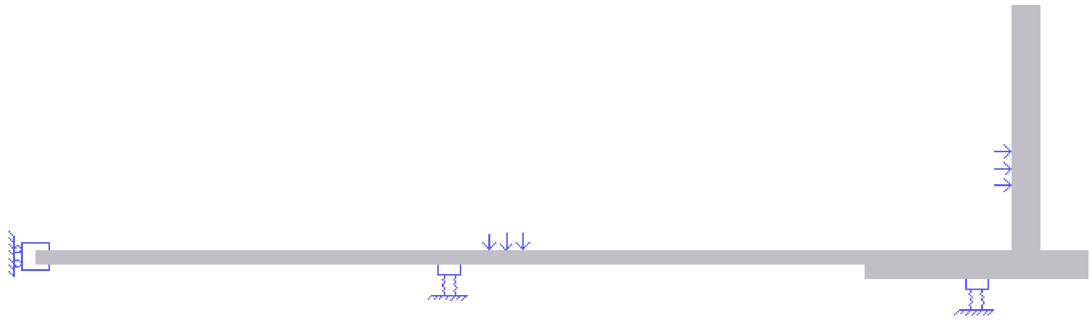


Figure 1.7: Model of the tank with boundary conditions included.

1.2.3 Postprocessing and analysis of results

The following figures, show the results from the simulation. The deformation of the structure is also included.

For the case of the horizontal displacements, Figure 1.8, we see that the part of the structure that undergoes a maximum horizontal displacement is the part located somewhat in the middle of the side wall. This makes sense as the pressure of the water inside the tank makes the wall to bend, inducing some curvature.

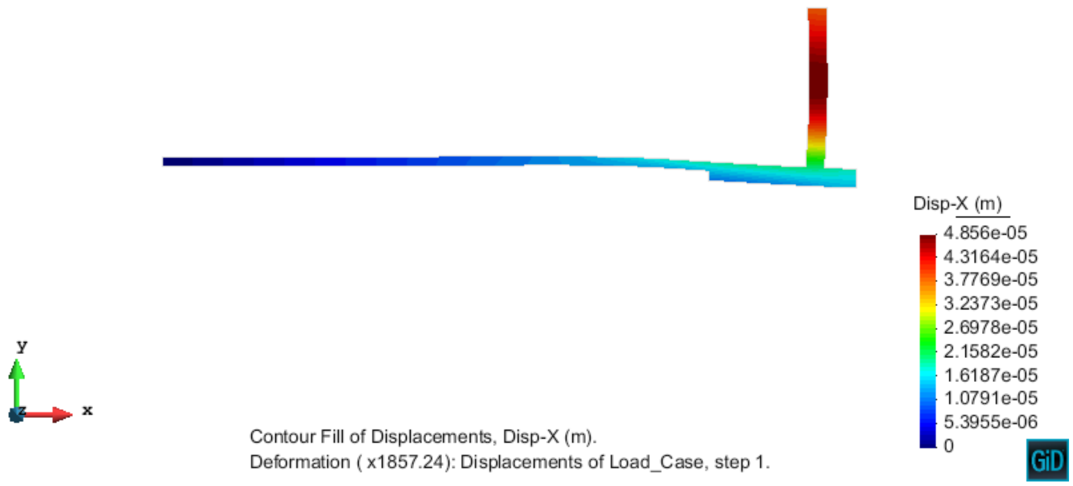


Figure 1.8: Displacements in the y direction.

As we can see in 1.9, maximum displacements (negative) are located within the right end part of the structure, the one furthest to the axis of symmetry. As expected, the structure tends to get curved from the sides, as a result of the pressure loading that water ejects on the concrete walls. This is also the part where more concrete is located. Note that had we not considered the self-weight of the structure, maximum displacements will be located in points closest to the axis of symmetry.

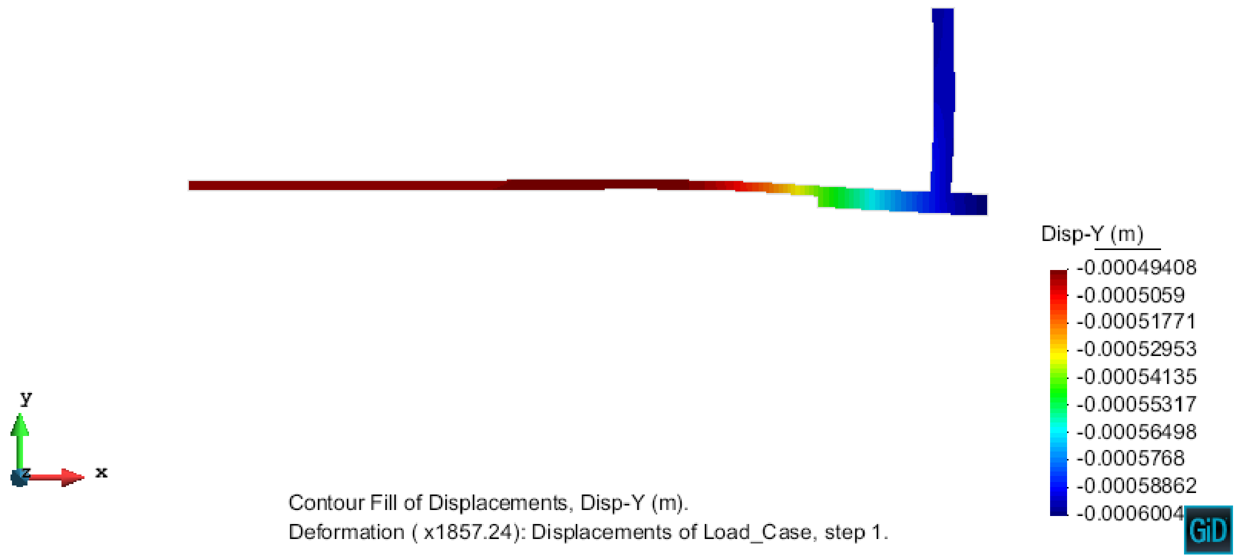


Figure 1.9: Displacements in the y direction.

Figure 1.10 down below shows the distribution of σ_x in the structure. It is noticeable to realize that maximum compression stresses are located within a sharp corner, which represents an abrupt change in the geometry.

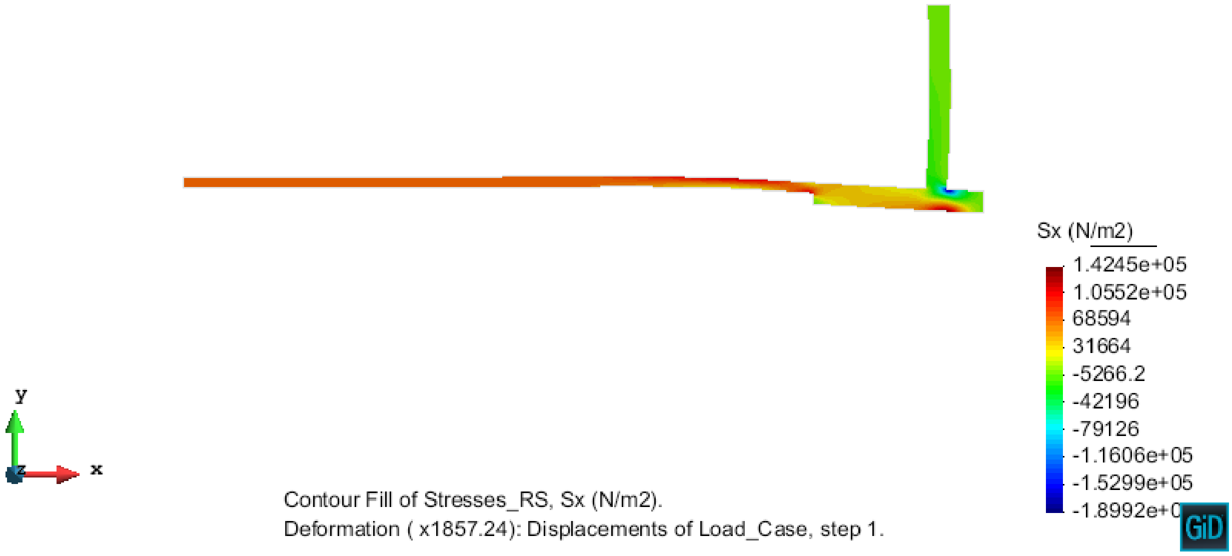


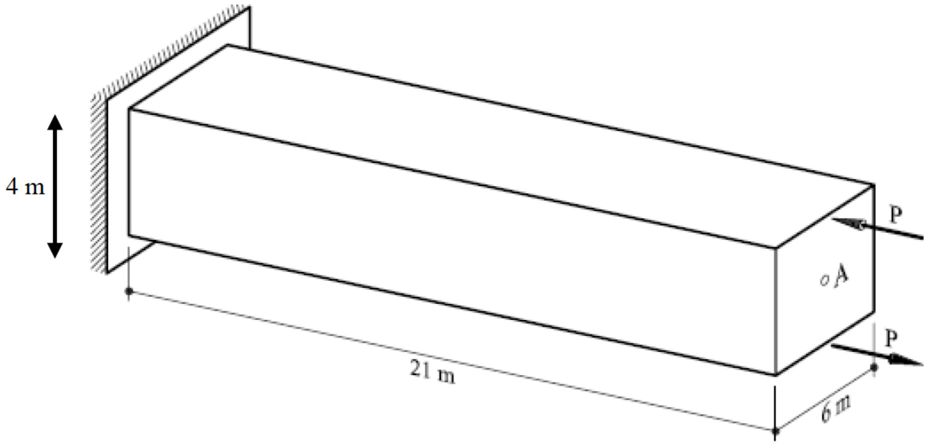
Figure 1.10: Diagram of the stresses σ_x on the tank.

For the case of the stresses σ_y, we obtain Figure 1.11

Figure 1.11: Diagram of the stresses σ_y on the tank

As already stated previously, the compressive water pressure on the wall tend to bend it, deriving into a curved shape of the right-hand side of the structure. As a result, maximum compression and tension stresses are located within this area. All in all, results are somewhat similar to the ones we obtained for the tank in the previous practice but it is important to realize that here we are not analyzing a plane strain state of a symmetric structure, instead we are analyzing a solid of revolution, where new effects such as the hoop stress will appear.

2 Analysis of the flexion of a beam with hexahedra elements



2.1 Purpose of the example

In this example, we aim to analyze the cantilever beam on the previous figure and compare the results with the traditional beam theory.

2.2 Analysis

2.2.1 Preprocessing

Geometry
First of all, we define the geometry of the beam in the preprocessor of GiD.

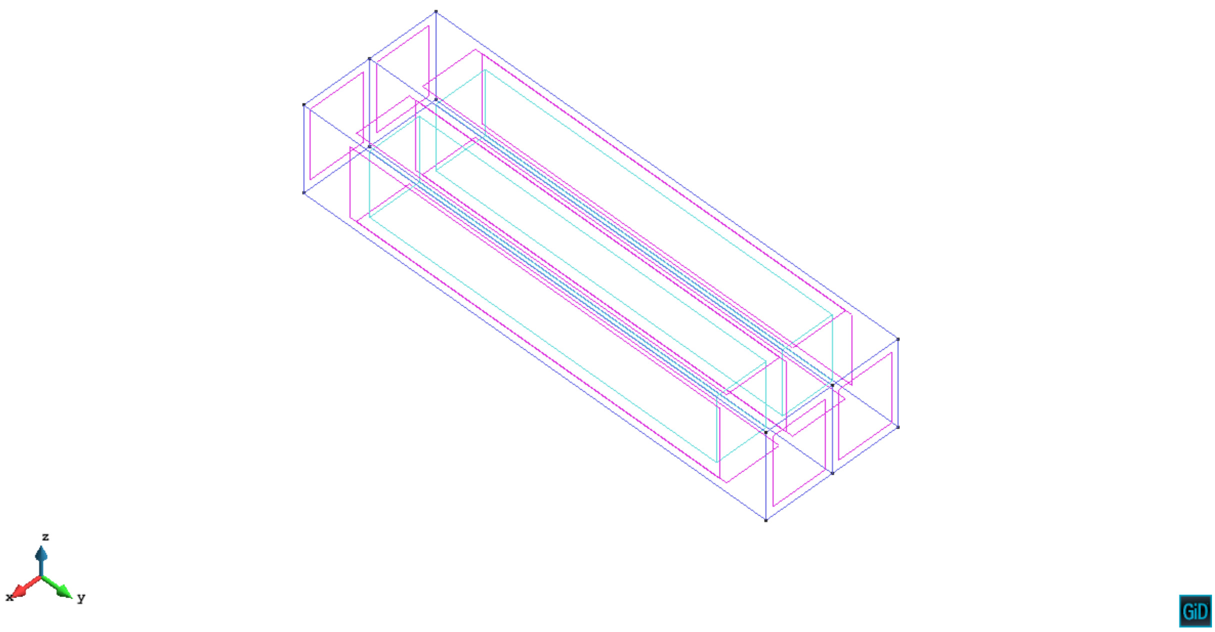


Figure 2.1: Geometry of the beam to analyze.

Data

Problem type

Once the geometry is defined, we can see which type of problem must be solved. In this case we have a 3D beam, then we face a 3D problem. Therefore we choose the module

$$Data/ProblemType/Ramseries\_Educational\_2D/3D\_Solids$$

Boundary Conditions

The types of boundary conditions that are enforced in this problem are the following:

- Displacements Constraints / Surface Constraints: we completely fix the end surfaces labeled as 2 and 1 in Figure 2.1 which are in one side of the beam.

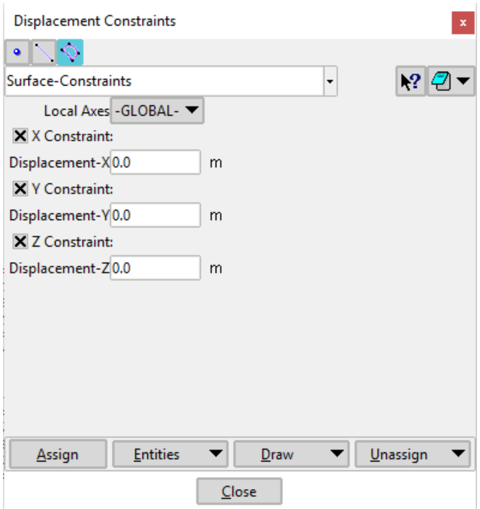
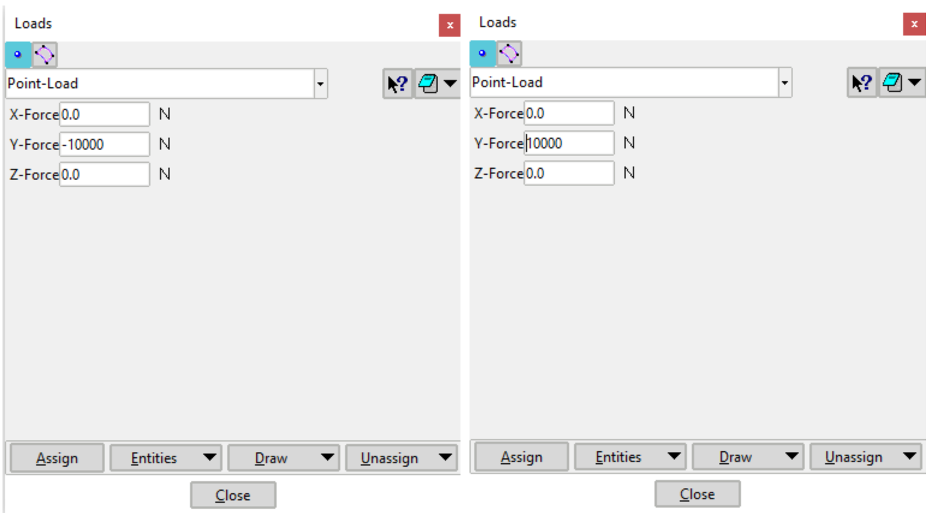


Figure 2.2: Definition of displacement boundary condition on the beam.

- Loads / Point-Load: We need to apply two point loads exactly on points 9 and (labeled in black in Figure 2.1) with values of 10000 N and -10000 N respectively.



(a) (b)

Figure 2.3: Loading definition in GiD.

Material

We use material with the following mechanical characteristics.

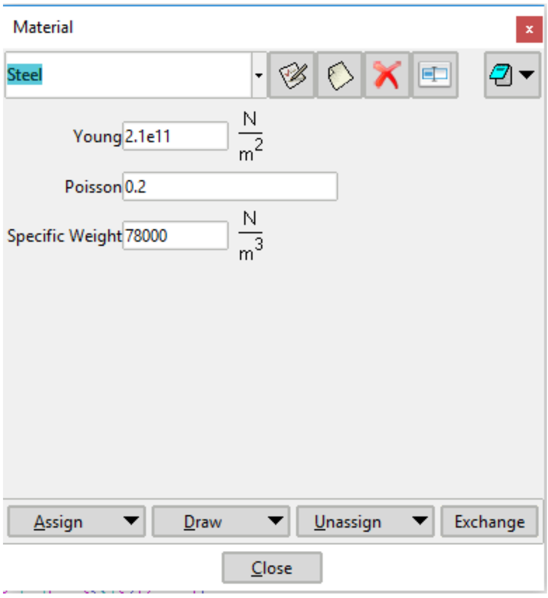


Figure 2.4: Material properties for exercise # 2.

Problem Data

In this section we specify some data necessary for the analysis:

- Problem title: Exercise 2
- ASCII output: NO
- Consider self-weight: NO
- Scale Factor: 1.0
- Results units: N-m-kg

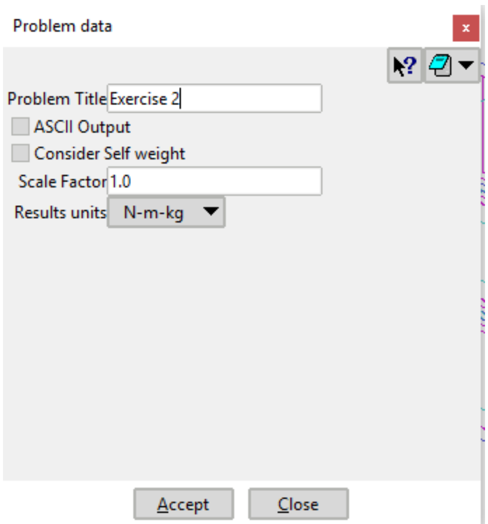
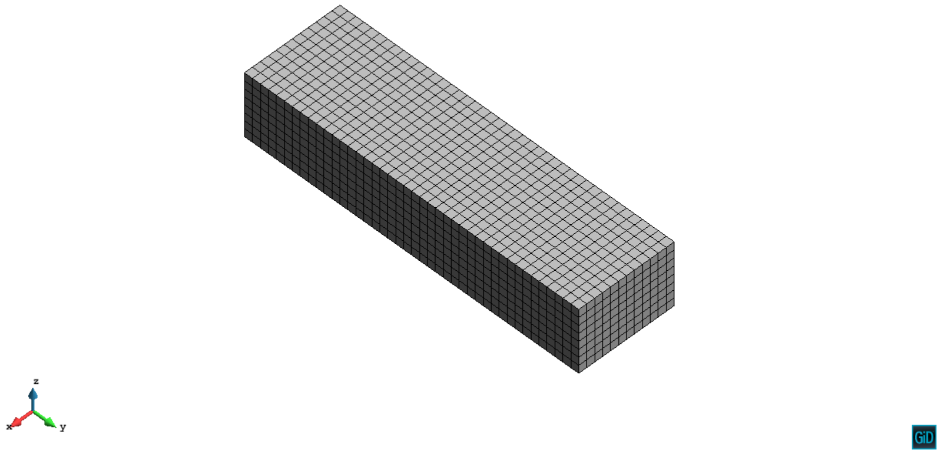


Figure 2.5: Problem data definition for exercise # 2.

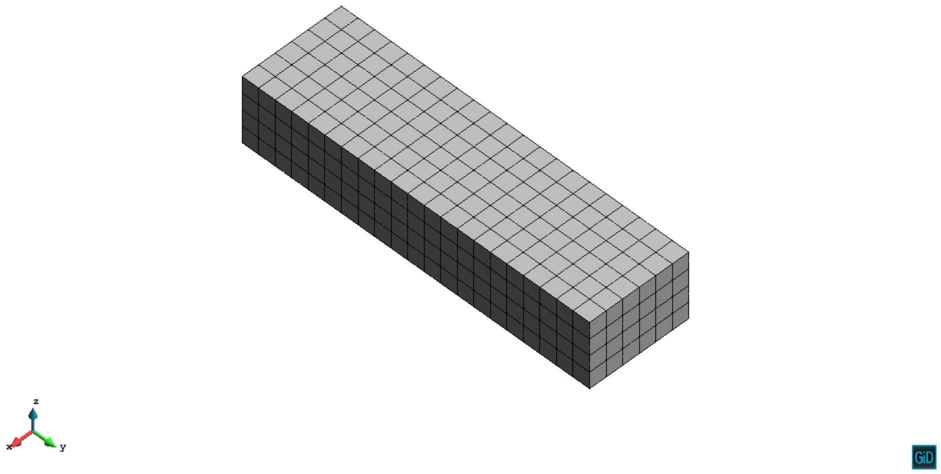
Mesh

We consider the following definitions of the mesh:

- Structured: Both meshes are structured
- Element type: Hexahedra
- Nodes: We use two meshes, one with linear element (8 nodes) and the other with elements with 20 nodes.



(a) Mesh with hexahedra elements with 8 nodes



(b) Mesh with hexahedra elements with 20 nodes

2.2.2 Processing

Once the pre-process stage is complete, we can proceed to run the problem. Figure down below, shows the problem we need to solve,

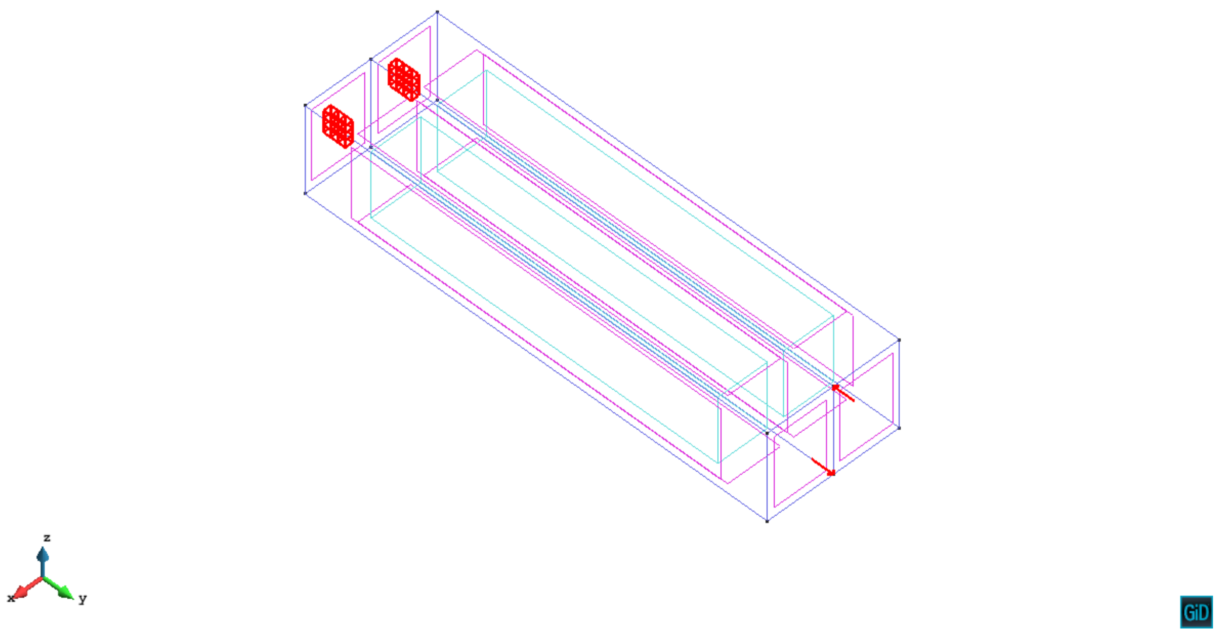


Figure 2.6: Final problem to solve. Boundary conditions are included.

2.2.3 Postprocessing and analysis of the results

The following figures show the results of the analysis obtained for exercise # 2. We present the stresses for both elements and also displacements in the z direction, the most representative ones. We have considered different meshes with eventually more nodes in order to lately compare with the analytical result.

As expected, the highest values of the vertical displacements are located in the free end of the beam, where the moment is applied. First, we present images for the 8-noded element.

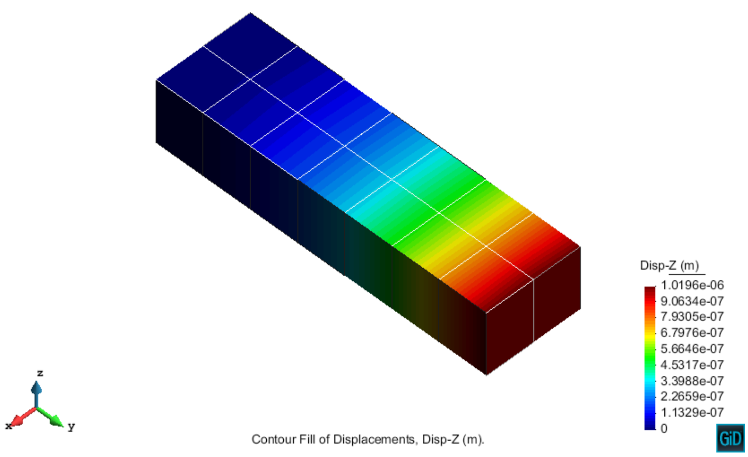


Figure 2.7: Displacements in the z direction for the linear hexahedral element with a coarse mesh (48 nodes).

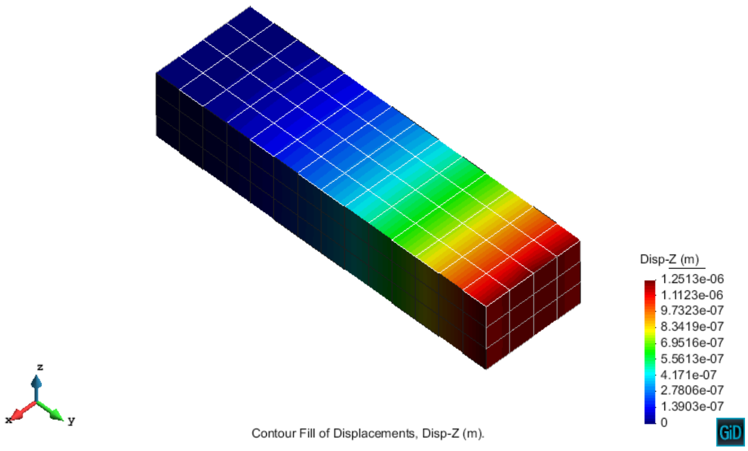


Figure 2.8: Displacements in the z direction for the linear hexahedral element for a mesh of 300 nodes.

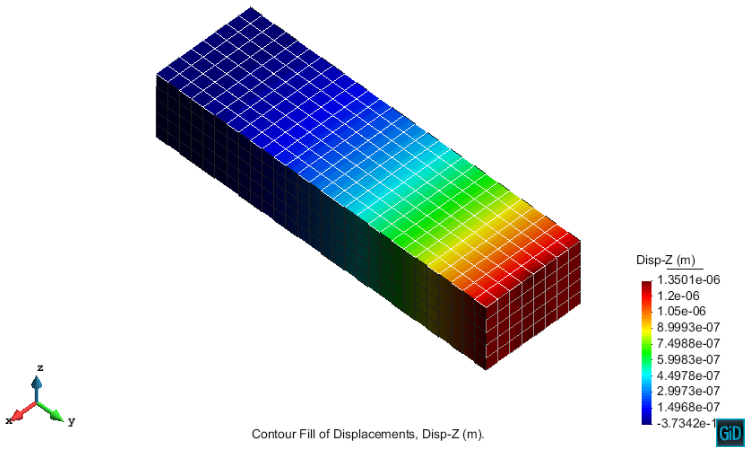


Figure 2.9: Displacements in the z direction for the linear hexahedral element for a mesh of 1566 nodes.

In the case of the stresses, it is clear that they concentrate nearly the area where the two nodal forces were applied.

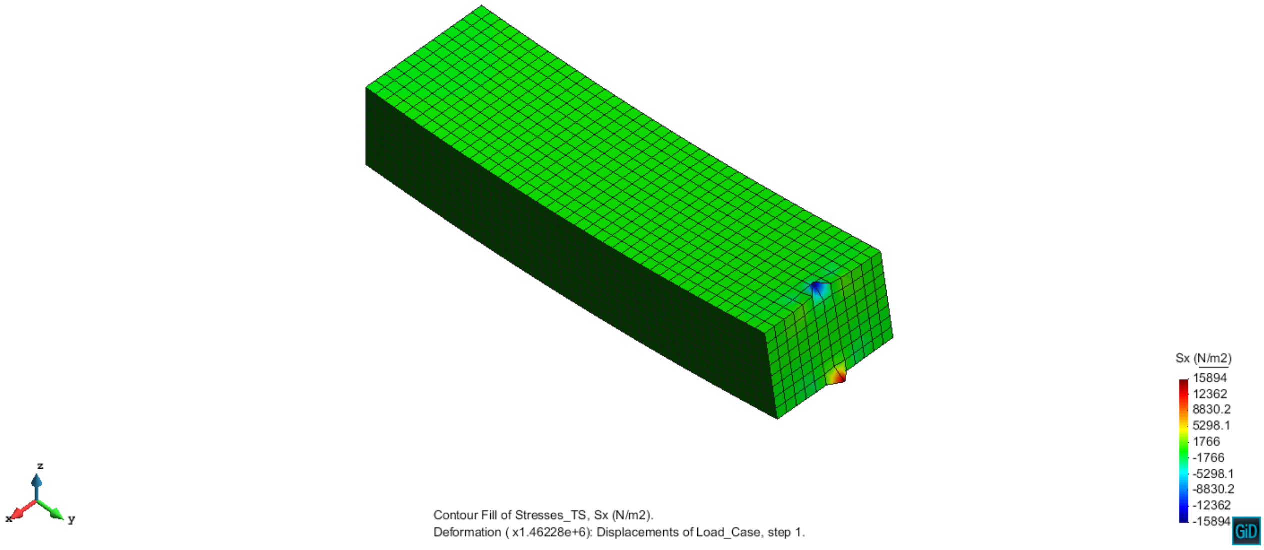


Figure 2.10: Stress component σ_x for the linear element.

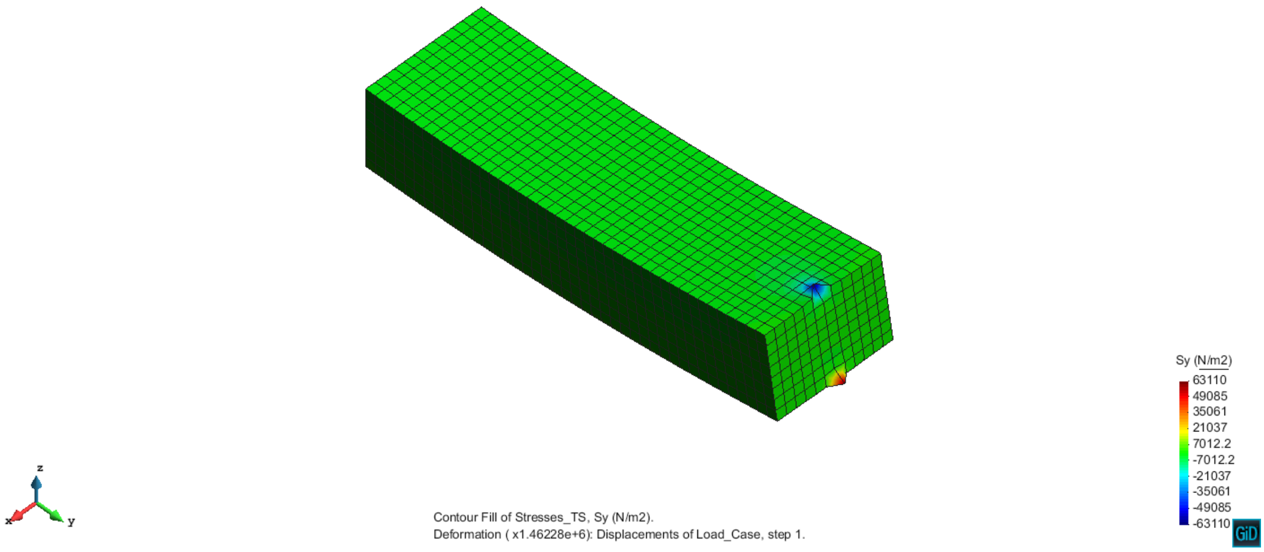


Figure 2.11: Stress component σ_y for the linear element.

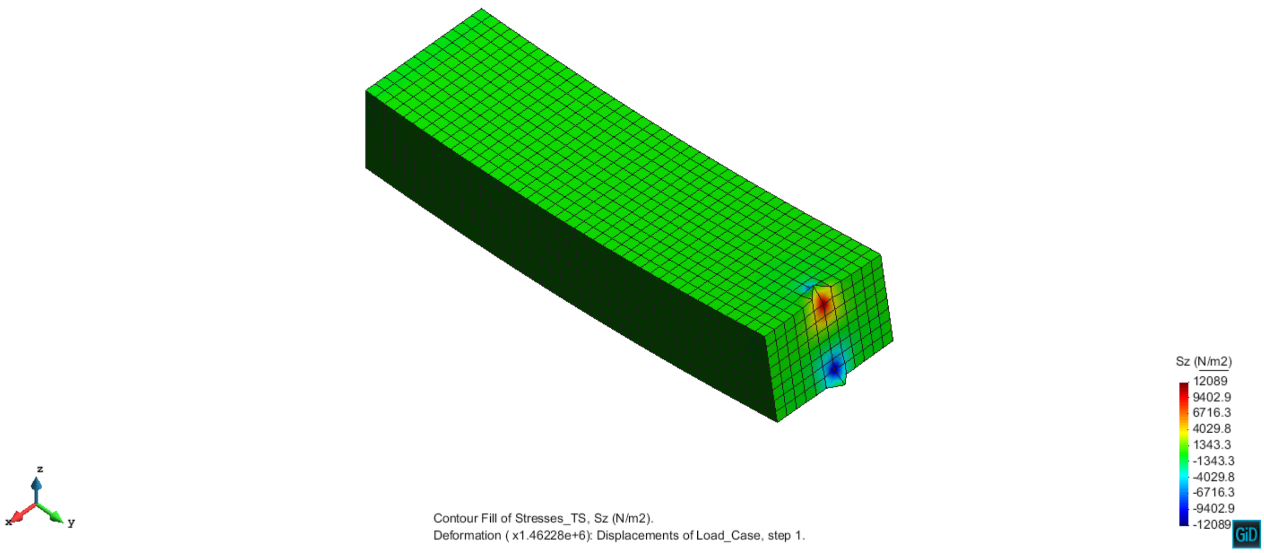


Figure 2.12: Stress component σ_z for the linear element.

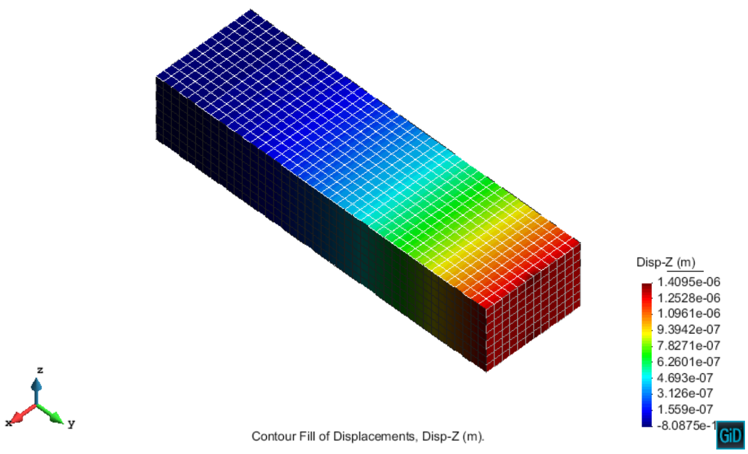


Figure 2.13: Displacements in the z direction for the linear hexahedral element for a mesh of 5031 nodes.

Next figures, present the results for the 20-noded hexahedral element.

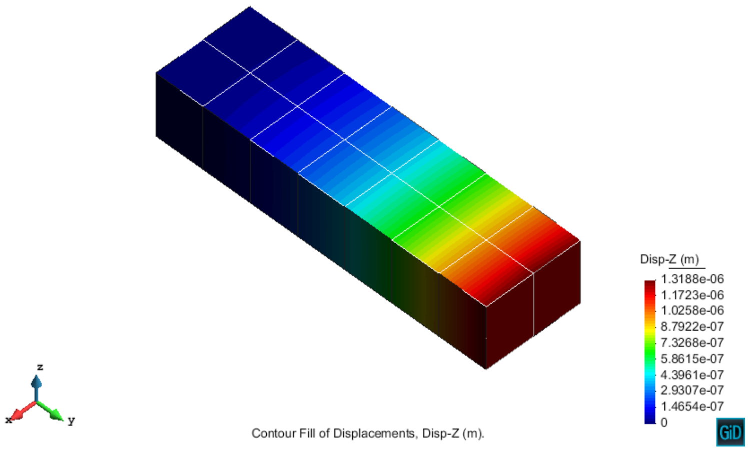


Figure 2.14: Displacements in the z direction for the quad hexahedral element for a coarse mesh of 146 nodes.

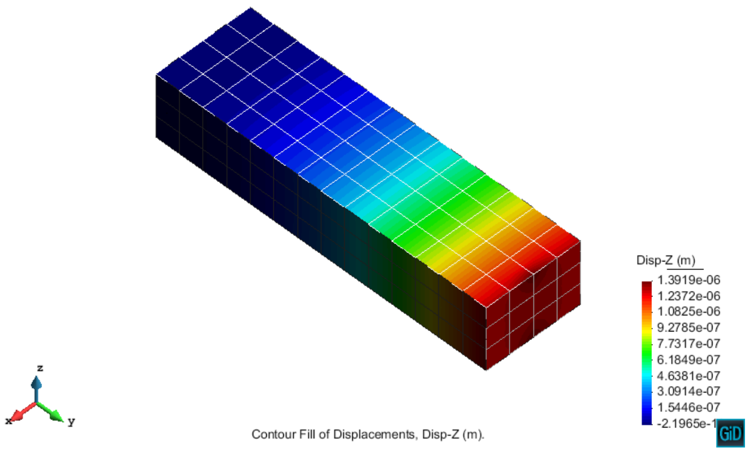


Figure 2.15: Displacements in the z direction for the quad hexahedral element for a mesh of 1045 nodes.

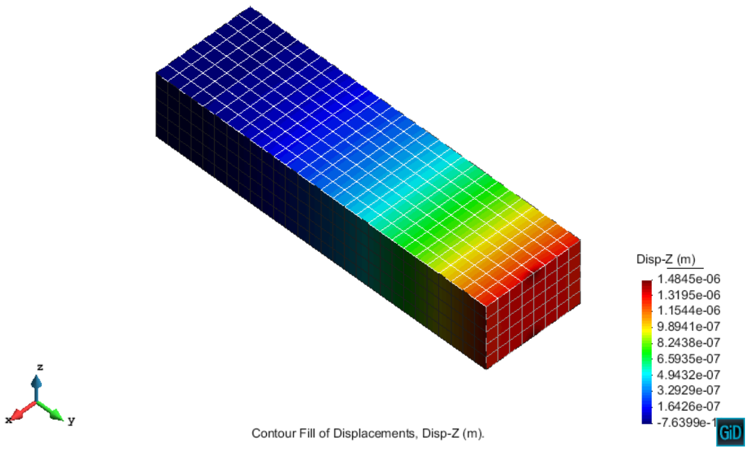


Figure 2.16: Displacements in the z direction for the quad hexahedral element for a mesh of 5775 nodes.

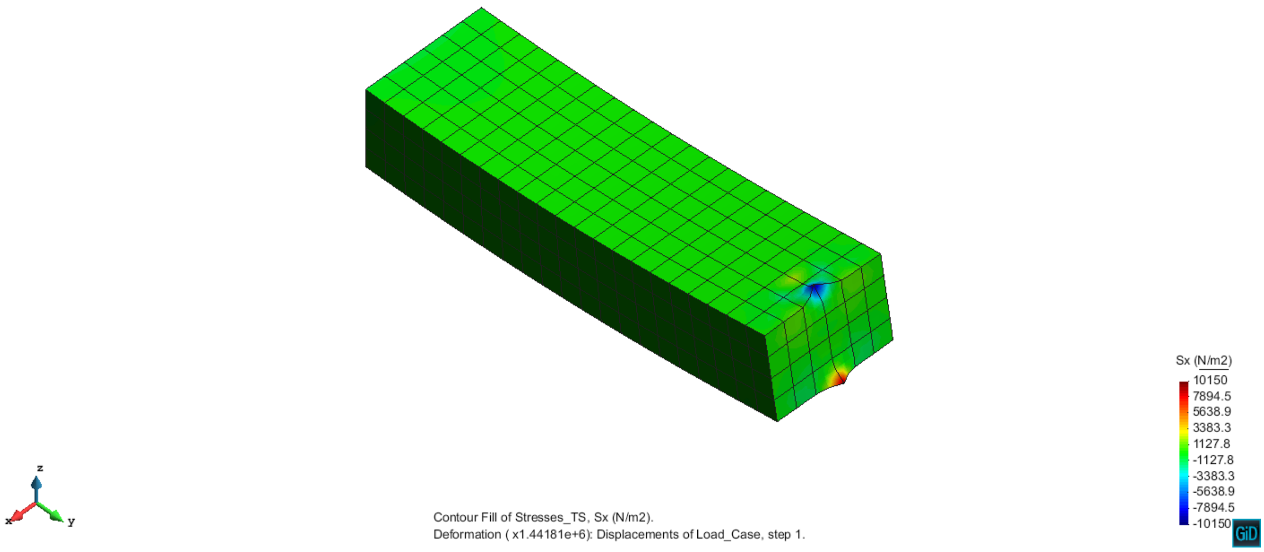


Figure 2.17: Stress component σ_x for the quad element.

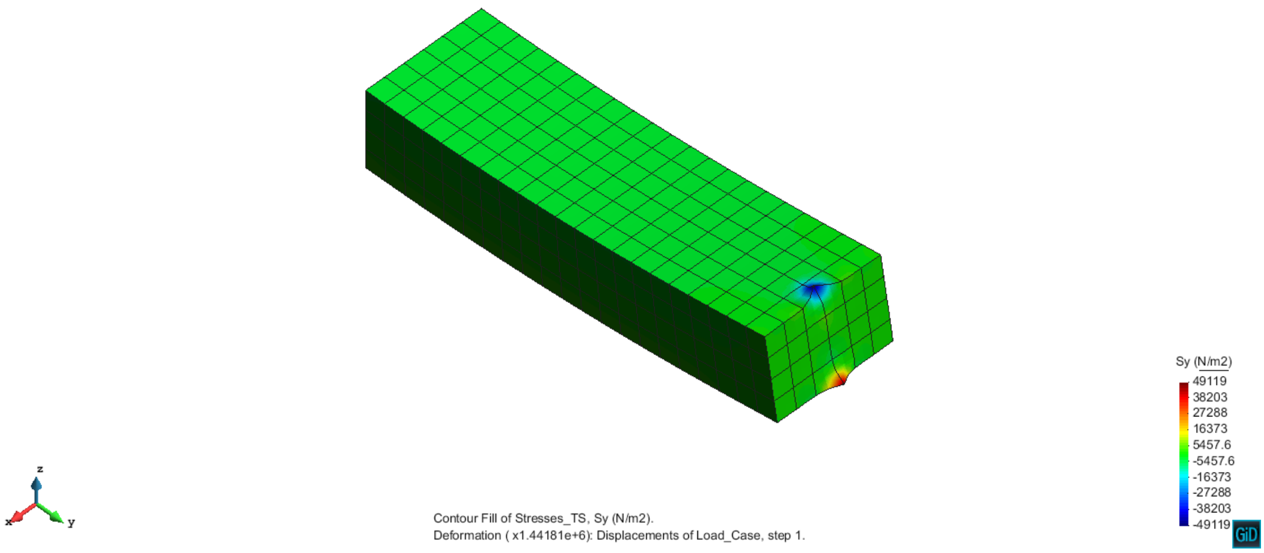


Figure 2.18: Stress component σ_y for the quad element.

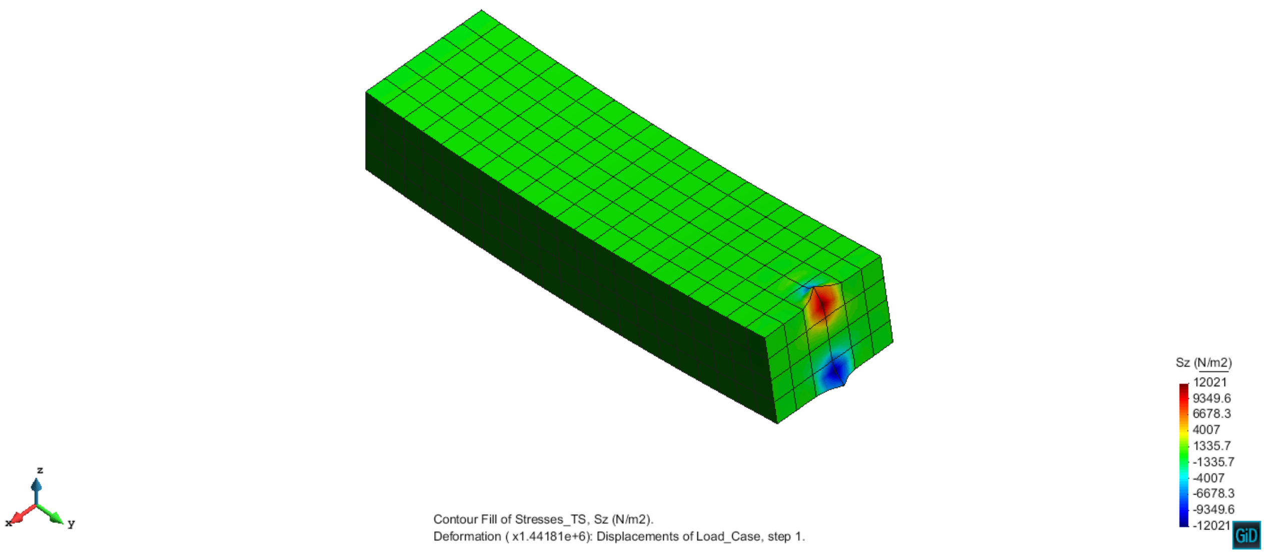


Figure 2.19: Stress component σ_z for the quad element.

For this problem, we can easily calculate the analytical value of the displacement at the point of the free end where the moment is applied. For this case we have that,

$$u_z|_M = \frac{ML^2}{2EI} = \frac{PhL^2}{2EI} = \frac{PhL^2}{2E\frac{bh^3}{12}} = \frac{6PL^2}{Ebh^2}$$

where the $|_M$ means where the moment is applied. Thus, taking into account that $P =$ 10000N, $h = 4$m, $L = 21$m, $E = 2.1e11 N/m^2$ and $b = 6$m, we obtain

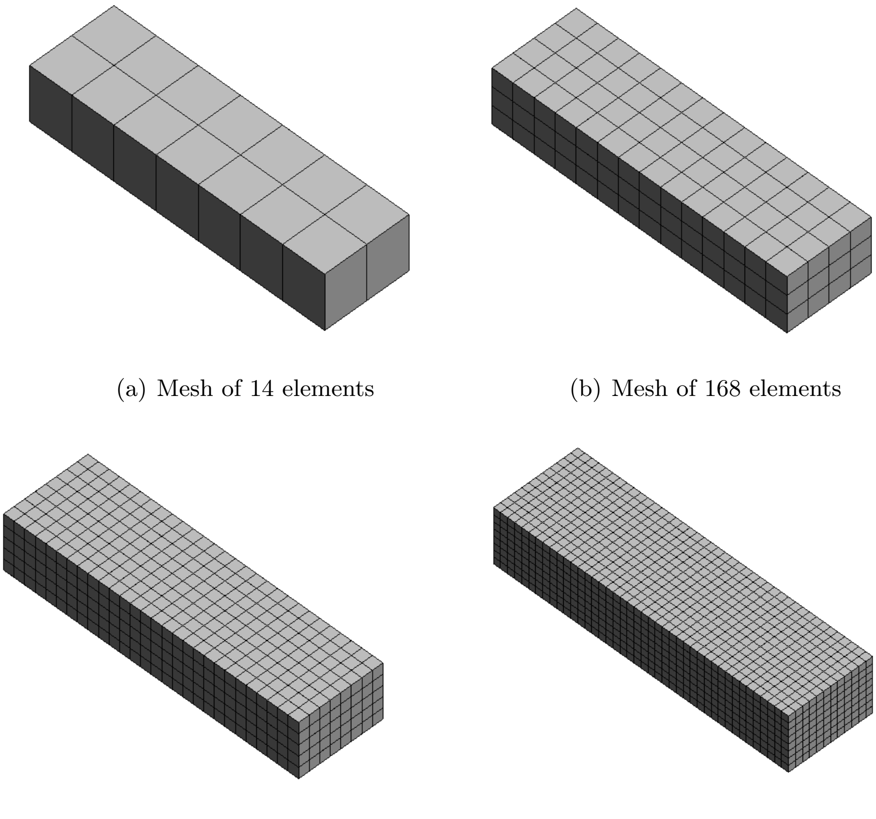
$$u_z|_M = 1.3125e-6 \quad m$$

In Table 2.2.3 we show a comparison of the displacements calculated for the 8-noded hexahedral element and the 20-noded hexahedral element for different meshes and the analytical value. Last column shows the relative error.

	Mesh information		
Element type	Nodes	GiD u_z [m]	Error %
Hexahedral element 8 nodes	48	1.0196e-06	22.32
	300	1.0591e-06	19.46
	1566	1.079e-06	15.59
	5031	1.3047e-06	0.59
Hexahedral element 20 nodes	146	1.3006e-06	0.15
	1045	1.3150e-06	0.19
	5775	1.3148e-06	0.17

Table 2.1: Comparison of different values of vertical displacement given in GiD for different meshes and the one from the analytical computation.

Figures down below show the meshes used for the comparison. [1]



(a) Mesh of 14 elements

(b) Mesh of 168 elements

(c) Mesh of 1120 elements

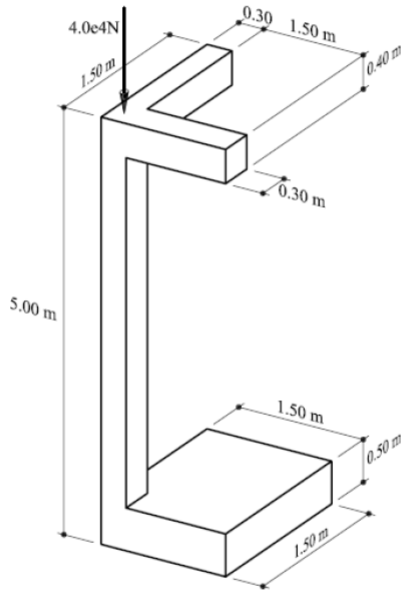
(d) Extra mesh used in the linear case with 4032 elements

Figure 2.20: Meshes considered for the comparison of numerical and analytical result for the displacements in the vertical direction.

[1] Here, we were not able to compute solutions with finner meshes because GiD always came up with an error of the type *"Error: Found EOF too early"*. We do not know why this error came up so Figure 2.20 (d) , is the finest mesh we were able to work with.

We observe that the quadratic element, since it has more nodes in the discretization, gives way better solutions even for quite coarse meshes. But, we also realize that the second mesh considered, even though it has quite more number of nodes, gives slightly a higher error. This might be due to some rounding error or instability of GiD in the computations near the solution. In the case of the linear element, quite coarse meshes give a huge error in the displacement. As we can see, a mesh with more than 5000 nodes is needed to get a reasonable result.

3 Foundation of a corner column



3.1 Purpose of the example

This model represents a corner column with its corresponding foundation. The most representative characteristic of this type of foundation is that the support reactions are eccentric with respect to the load to which the column is subjected. This translates into a flexion of the column and the lifting of the base slab.
The purpose of this exercise is to analyze the stress state of the column and the slab assuming that the slab is elastically supported by the ground and to determine whether or not the slab suffers from lifting.

3.2 Analysis

3.2.1 Preprocessing

Geometry

The initial step is to create the geometry using the preprocessor of GiD. In this case, in order to facilitate the assembly of the structure, an auxiliar sketch was created in the XY plane and then the different sections were extruded to their corresponding z coordinate.

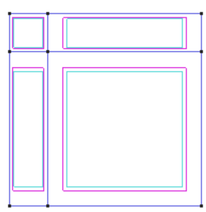


Figure 3.1: Auxiliar sketch in plane XY

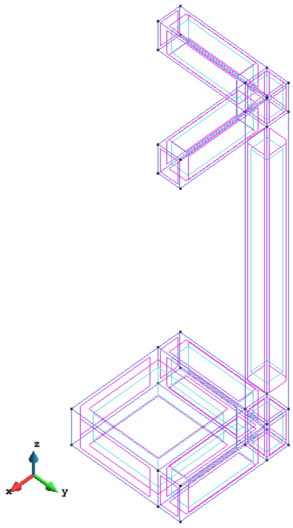


Figure 3.2: Isometric view of the structure

Data

Problem type

Then, the following step is to select the problem type that fits the model. Since this is a 3D structure, the '3D Solids' of the Ramseries Educational module is selected.

Boundary Conditions

The appropriate boundary conditions have to be defined in order to the problem to be solvable. In this exercise, several types of conditions need to be set up.

- Elastically supported ground: in order to simulate how the slab is supported by the ground, an elastic displacement constraint is defined for all the bottom surfaces.
- Restriction of XY plane translation: the structure is supposed to be attached to other symmetric structures at the top, so the end of the top beams are set to have zero displacement in the X and Y directions.
- Distributed load: the applied force is defined as a distributed load applied in the top surface of the column.

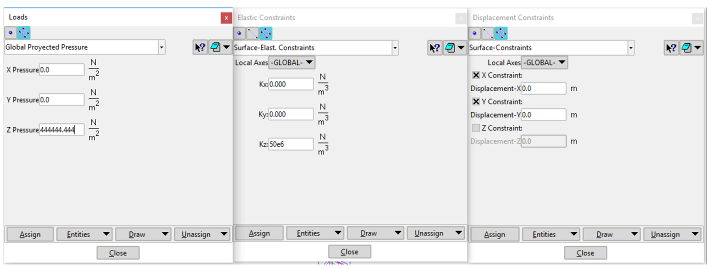


Figure 3.3: Definition of the boundary conditions

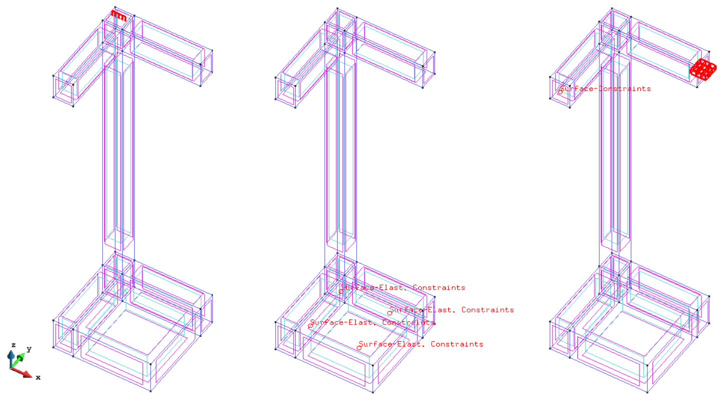


Figure 3.4: Visualization of the boundary conditions

Material

The structure is made out of concrete, so the correspoding properties are defined and the material is assigned to the whole model.

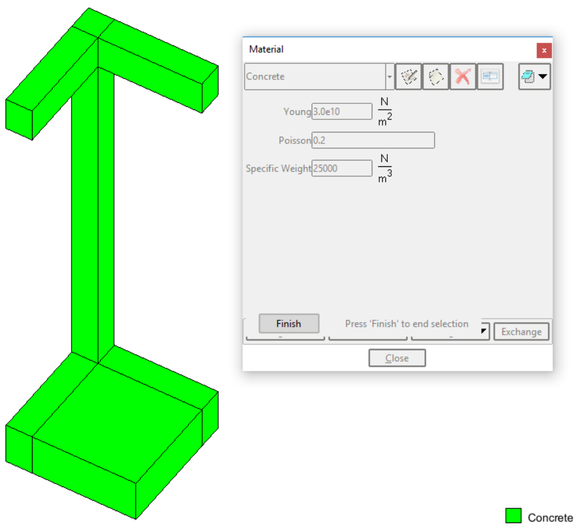


Figure 3.5: Material definition

Problem Data

In this section the remaining necessary data for the analysis has to be specified. For this problem:

- Problem title: Exercise 3
- ASCII output: NO
- Consider self-weight: NO
- Scale factor: 1
- Results Units: N-m-kg

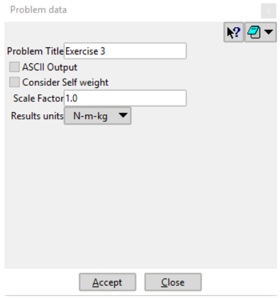


Figure 3.6: Problem data for Exercise 3

Mesh

The mesh was generated using the following options:

- Type: semistructured
- Element: hexahedral
- Order: linear (8 nodes)

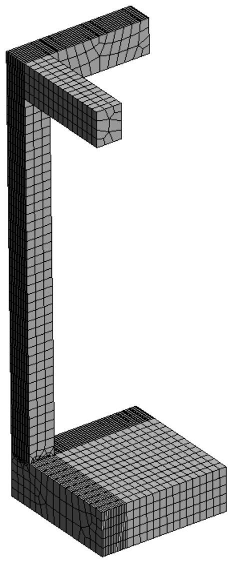


Figure 3.7: Semistructured mesh of 8-node hexahedrals

3.2.2 Processing

Once all the data is properly defined and the mesh is generated, the simulation can be run and the results analyzed.

3.2.3 Postprocessing and analysis of results

The following figures show the results obtained for the displacement of the foundation of a corner column. The most important result to be considered is the one in the Z direction, that allows to predict if the slab of the foundation will suffer from lift or not.

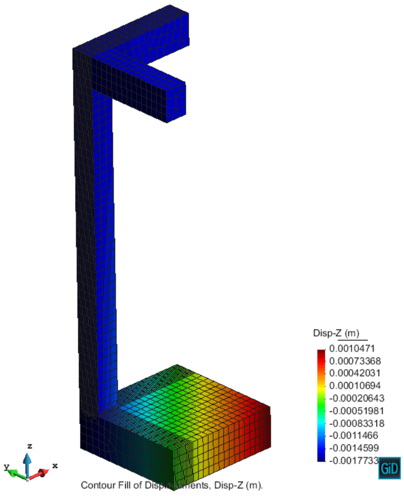


Figure 3.8: Displacement in the Z direction of the foundation

From Figure 3.8 it is possible to extract that the maximum displacement of the slab is produced at the corner opposite to the column, as expected, and its value is of order $10^{-3} \quad m$. Taking into account that the thickness of the slab is of order $1 \quad m$, it can be considered that the lifting of the slab is negligible.
In fact, this model does not consider the weight of the ground above the slab. If this weight is considered as a load distributed over all the surface, the lifting produced in the corner would be even lower.

The analysis also gives the distribution of stresses over the structure, shown in the next figure. As can be expected, the maximum stresses are obtained in the top bars, were the restrictions in displacements are applied and reaction forces are generated.

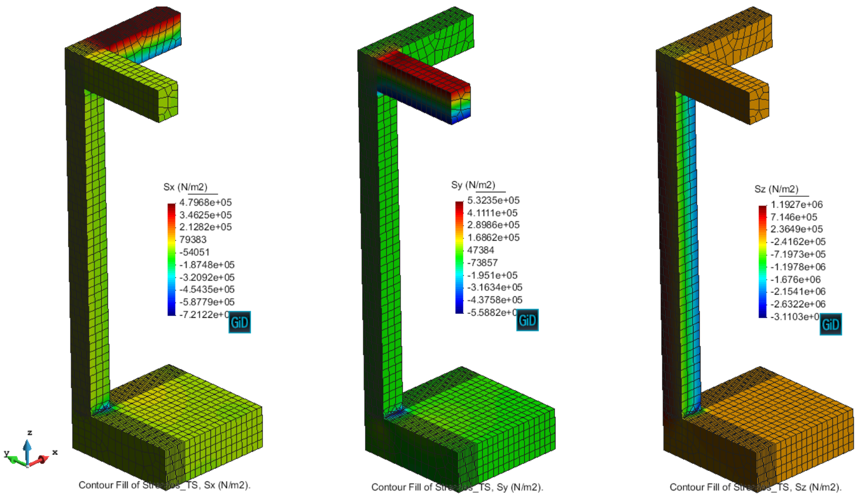


Figure 3.9: Stress distribution of the foundation