UNIVERSITAT POLITÈCNICA DE CATALUNYA, BARCELONA

MSC. COMPUTATIONAL MECHANICS ERASMUS MUNDUS

GID ASSIGNMENT 2

Computational Structural Mechanics & Dynamics

Author:
Nikhil Dave

Date: March 31, 2018

Exercise 1: Circular tank

The figure shows a circular tank made of reinforced concrete. It is used for the storage of water in a water purification plant. Analyse the structural behaviour of the tank. Use quadrilateral elements with four nodes.

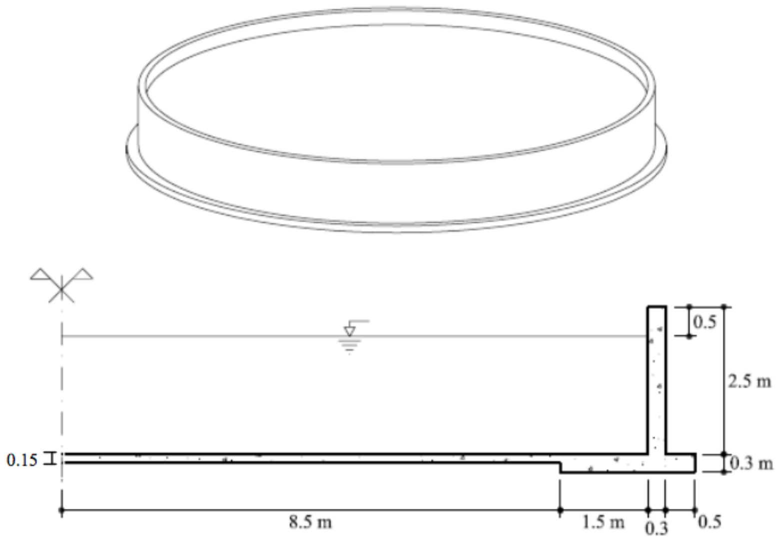


Figure 1: Circular tank

Solution:

1.1. Purpose of the exercise

The objective of this exercise is to analyse the structural behaviour of a circular water tank.

1.2. Analysis

1.2.1 Pre-processing

(i) Geometry

The first step of pre-processing is to model the geometry as per the given dimensions in GiD as shown in Figure 2.

Figure 2: Defining the geometry

(ii) Data
Once the geometry is defined, we apply the given data to the model.

Problem type

Since, the given problem is axisymmetric, we select the Rev Solids problem type from the Ramseries Educational module.

Boundary Conditions

Next, we define the boundary conditions for the problem as shown in Figure 11. Firstly, displacement constraint is applied to prevent movement in x-direction to simulate symmetry condition. Also, an elastic constraint i.e. ballast coefficient is applied to the surfaces at the bottom. Additionally, to incorporate the effect of water in the tank, we consider the uniform load (weight) supported by the horizontal surface and linear load for the pressure exerted by water on the vertical surface.

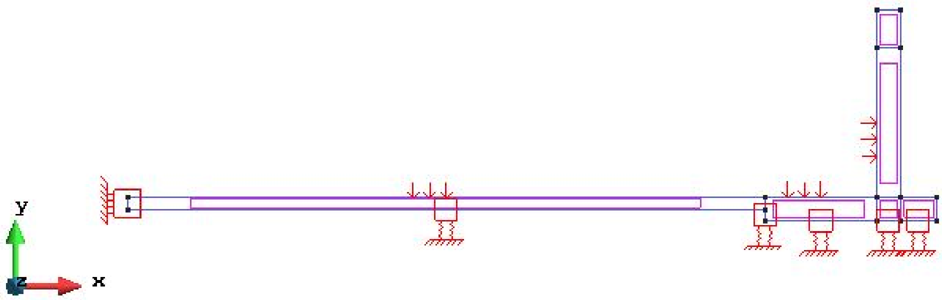


Figure 3: Boundary conditions

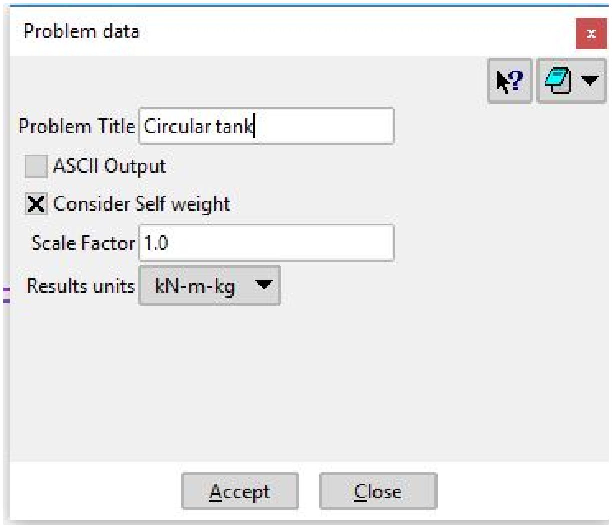
Material

The material properties of the structure are defined with the parameters of concrete as shown in Figure 4.

Problem Data

The definition of problem data is an important step for the analysis where the options like title, type of problem and the result units are to be selected. It is important to note here that the self weight is considered in this problem with a scale factor of 1.0. Figure 4 shows the data used for this problem.

(a) Defined material



(b) Problem data

Figure 4: Material and problem data defined

Mesh

In this problem, a structured mesh is used with linear quadrilateral elements with element size of 0.05 as shown in Figure 5.

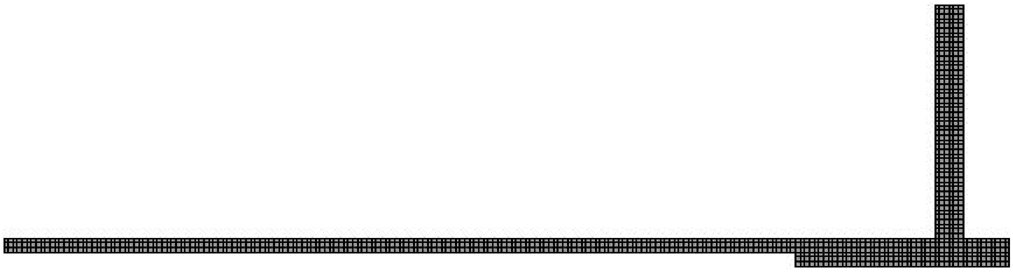


Figure 5: Structured mesh generation with linear quadrilateral mesh elements

1.2.2 Processing

In this section, we calculate the solution of the problem for the given data and generated mesh.

1.2.3 Post-processing

The results obtained for the problem are shown below. Firstly, the displacements results in the x and y-direction are plotted on the deformed configuration in Figures 6 and 7.

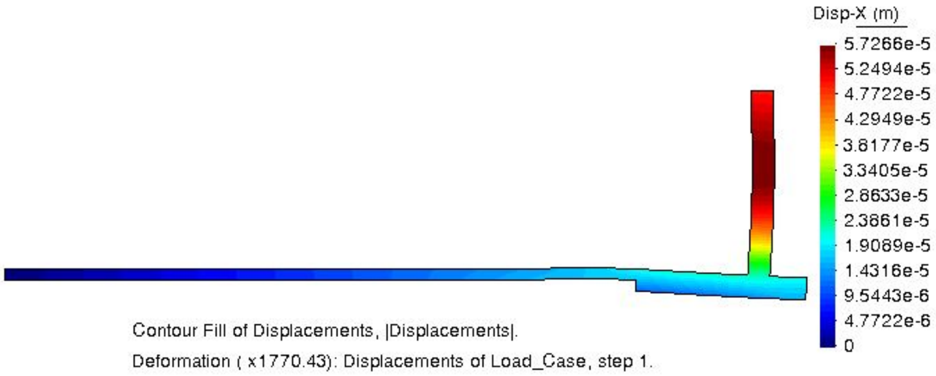


Figure 6: Results: displacements in the x-direction plotted on deformed configuration

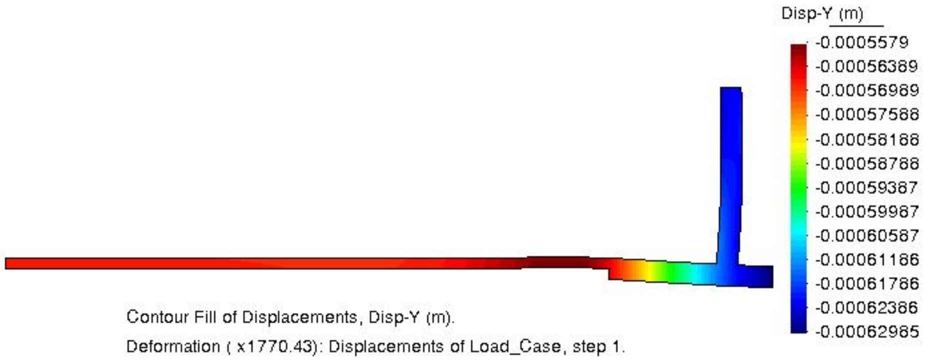


Figure 7: Results: displacements in the y-direction plotted on deformed configuration

This is followed by the result plots for the stresses in the x and y-direction shown in Figures 8 and 9.

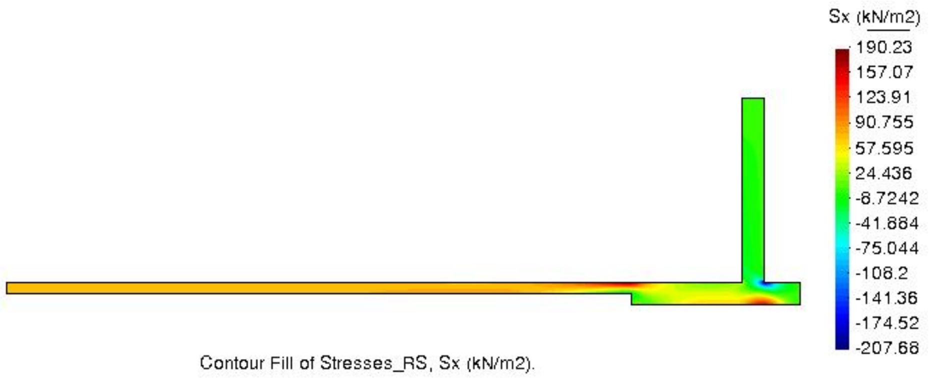


Figure 8: Results: Plot of stresses S_x

Figure 9: Results: Plot of stresses S_y

1.3. Comparison of the results

From the displacement plots, we see that maximum horizontal displacement is observed in the middle of the wall of the tank due to the bending effect caused by water pressure. The effect of linear loading applied to the wall plus the self-weight assumed can also be seen, in the stress plot (S_y), causing variations in the stress values as expected.

On the other hand, the bottom of the tank shows constant stress (S_y) due to the applied uniform loading. The wall behaves as a cantilever beam, similar to the one obtained for the plane strain tank we analysed in the previous assignment. Due to deformation, we have tension to the left and compression to the right side of the wall. Similarly, the floor shows tension on the top and compression on the bottom surface due to bending effect caused by loading and consequent soil reaction.

The results are closely related to the ones we obtained for the tank in the previous assignment, though it was a case of plane strain state of a symmetric structure and here we analysed a solid of revolution problem.

Exercise 2: Analysis of the flexion of a beam using hexahedra elements

Analyse the cantilever shown in the figure, submitted to the action of a moment at the far end. Compare the results obtained with the beam theory. Use hexahedra elements with 8 and 20 nodes.

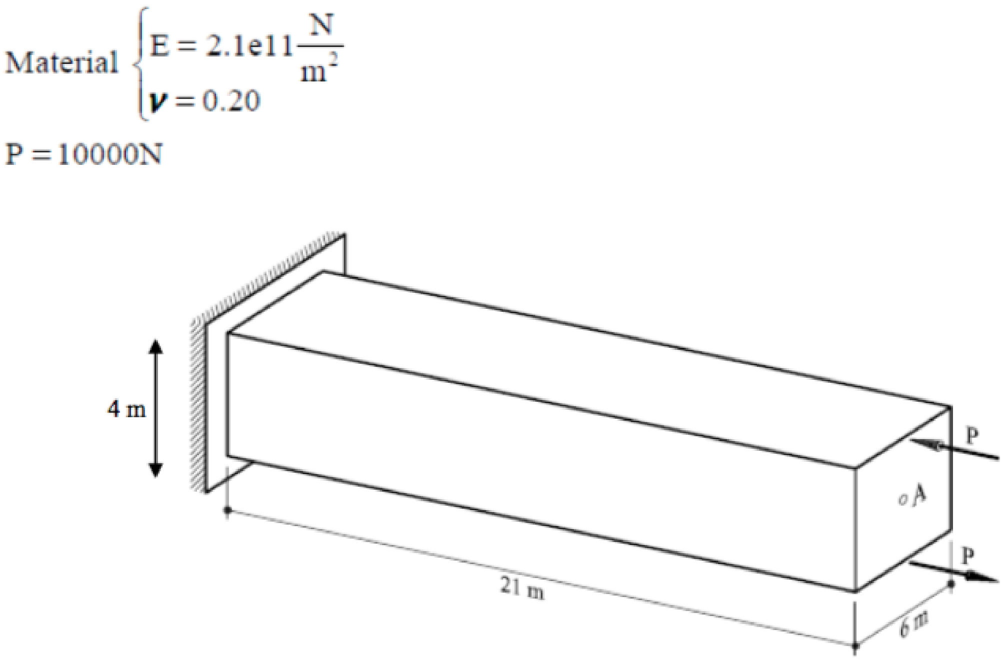


Figure 10: Cantilever beam

Solution:

2.1. Purpose of the exercise

The objective of this exercise is to analyse the given cantilever and compare the results with the traditional beam theory.

2.2. Analysis

2.2.1 Pre-processing

(i) Geometry

The first step of pre-processing is to model the geometry of the beam as per the given dimensions in GiD as shown in Figure 11.

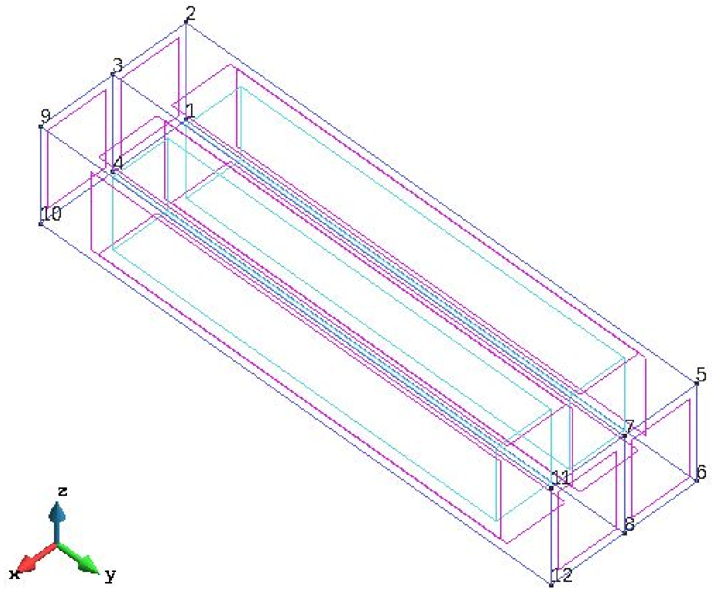


Figure 11: Defining the geometry

(ii) Data
Once the geometry is defined, we apply the given data to the model.

Problem type

Since, the given problem is 3D in nature, we select the 3D Solids problem type from the Ram-series Educational module.

Boundary Conditions

Next, we define the boundary conditions for the problem as shown in Figure 12. Surface constraint is imposed to fix one end of the beam. Two point loads are applied with equal and opposite values of 10000 N.

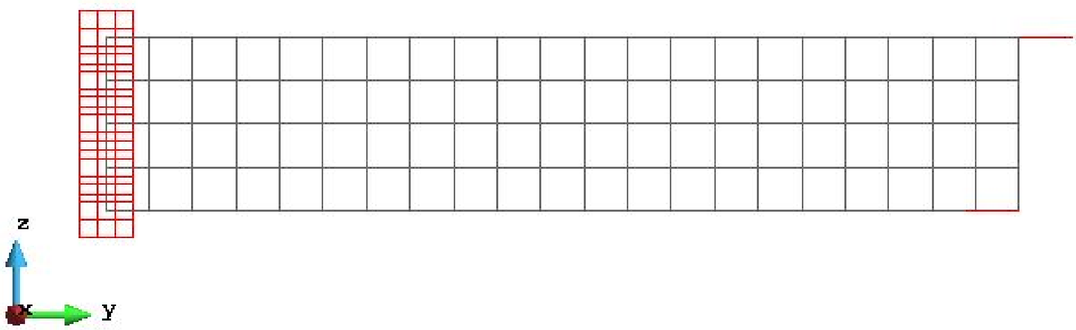


Figure 12: Boundary conditions in 2D view

Material

The material properties of the structure is defined with the given parameters shown in Figure 13.

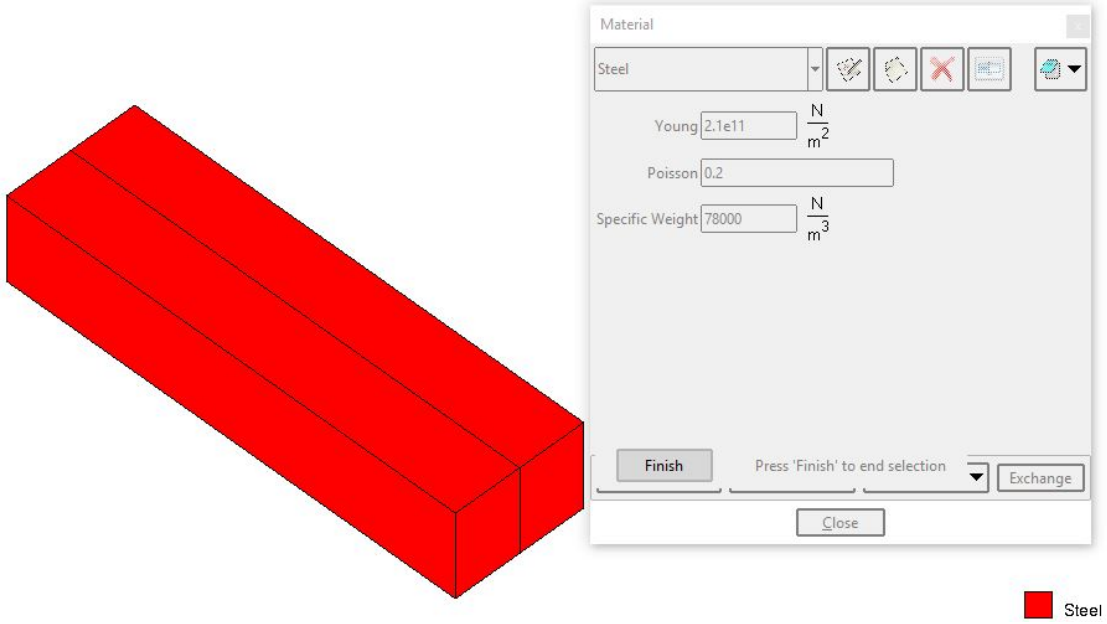


Figure 13: Material properties

Problem Data

Figure 14 shows the necessary problem data used for this problem.

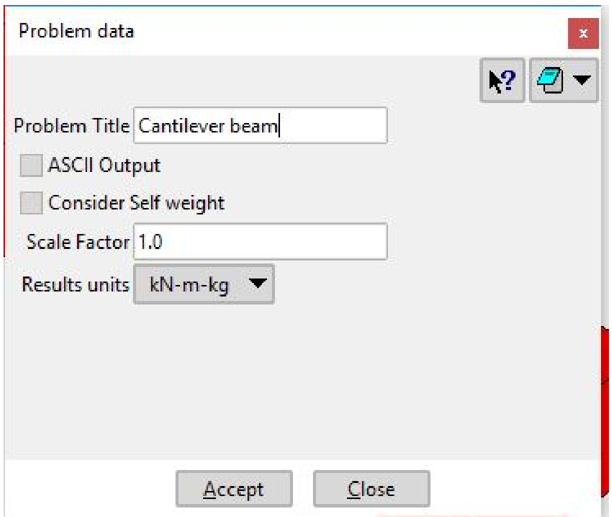
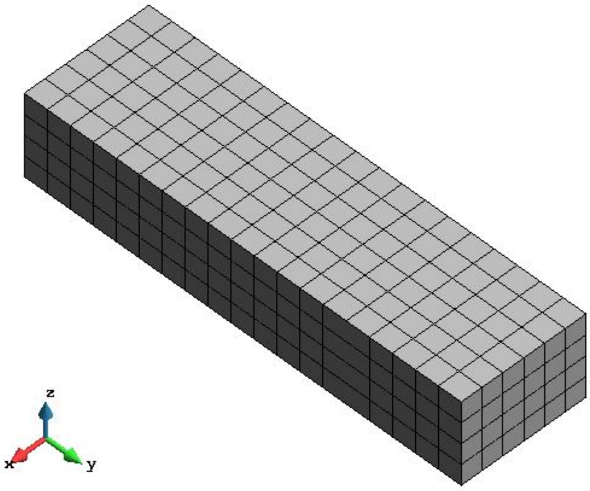


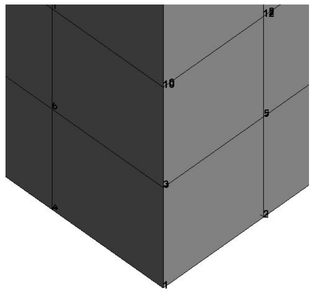
Figure 14: Problem data

Mesh

In this problem, we consider two types of structured hexahedral meshes - one with 8 nodes and the other with 20 nodes as shown in Figures 15 and 16

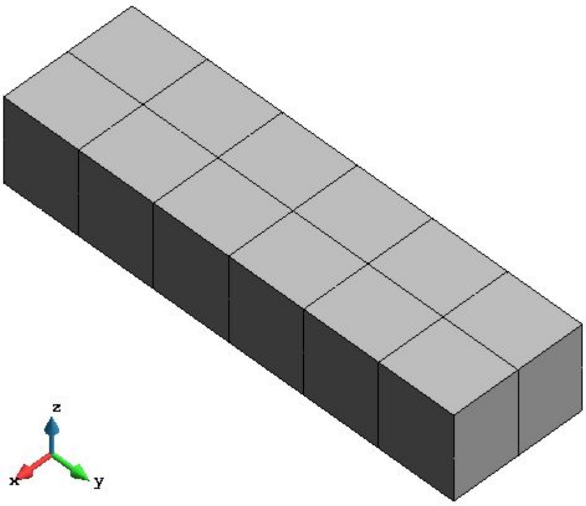


(a) Generated mesh for Case 1

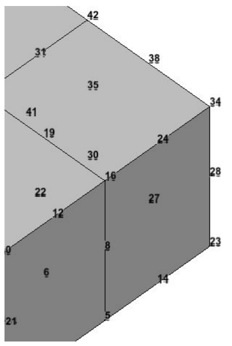


(b) 8 noded element labeled

Figure 15: Structured hexahedral mesh generated - 8 noded



(a) Generated mesh for Case 2



(b) 20 noded element labeled

Figure 16: Structured hexahedral mesh generated - 20 noded

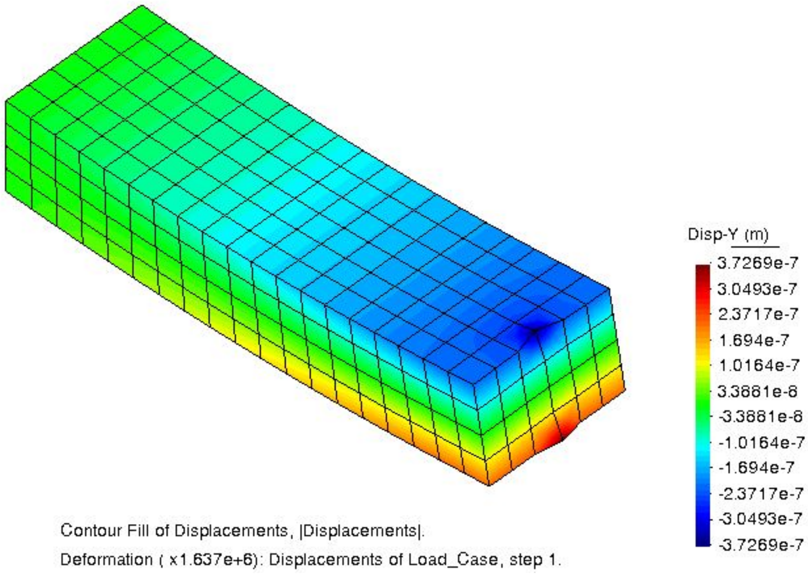
2.2.2 Processing

In this stage, we run the problem with the given data and using different meshes generated.

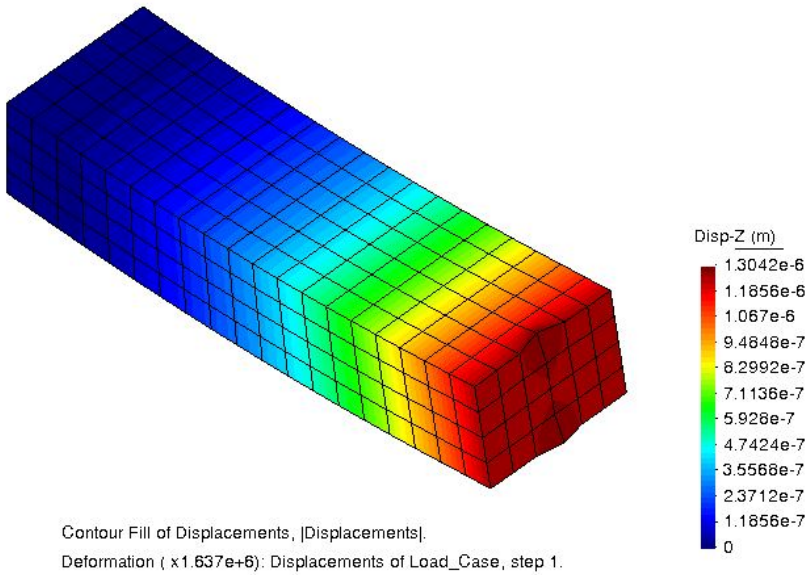
2.2.3 Post-processing

The results obtained for the deformation of the structure with 8 noded hexahedral elements are shown below:

Case 1: 8 noded hexahedral - 456 elements and 700 nodes



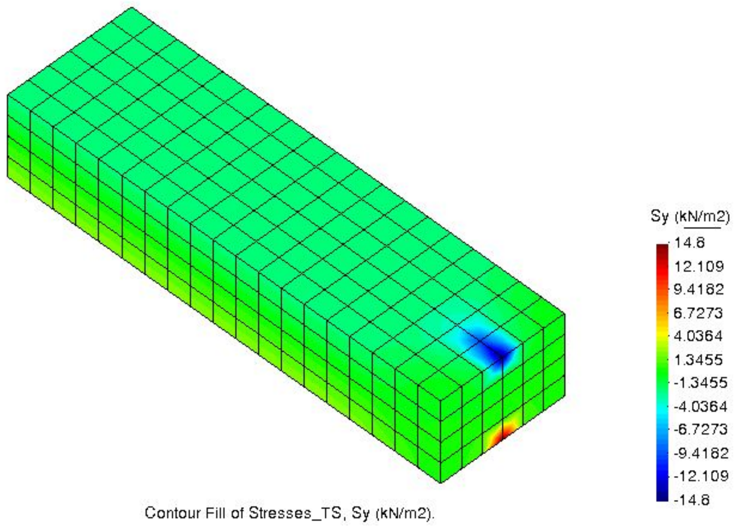
(a) y-displacement plot



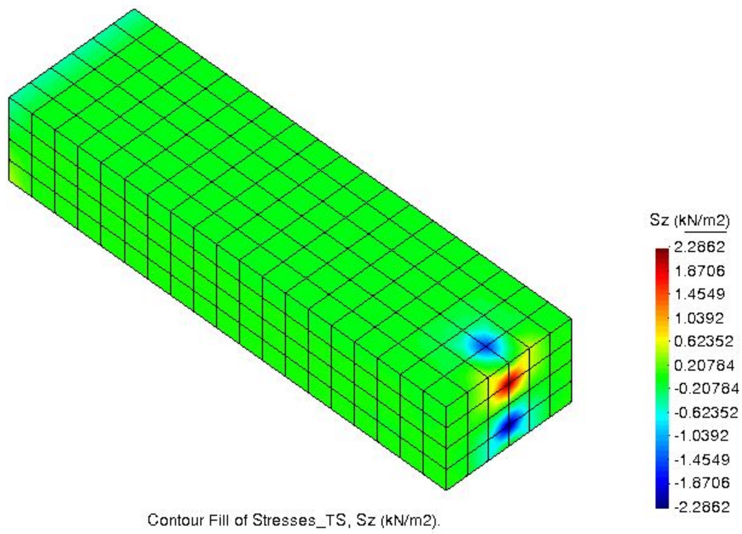
(b) z-displacement plot

Figure 17: Results: Displacement plot in the deformed configuration

This is followed by the stress distribution plots obtained for this case. It can be easily inferred from the results obtained that the the nodes where the point loads are applied, undergo maximum deformation and also are the stress concentration zones. To visualise this, Figure 19 shows the deformed configuration with the maximum z-displacement values.



(a) σ_y plot



(b) σ_z plot

Figure 18: Results: Stress distribution plot

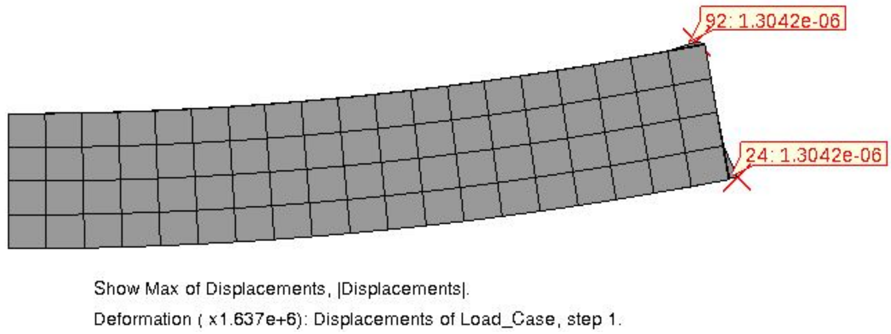
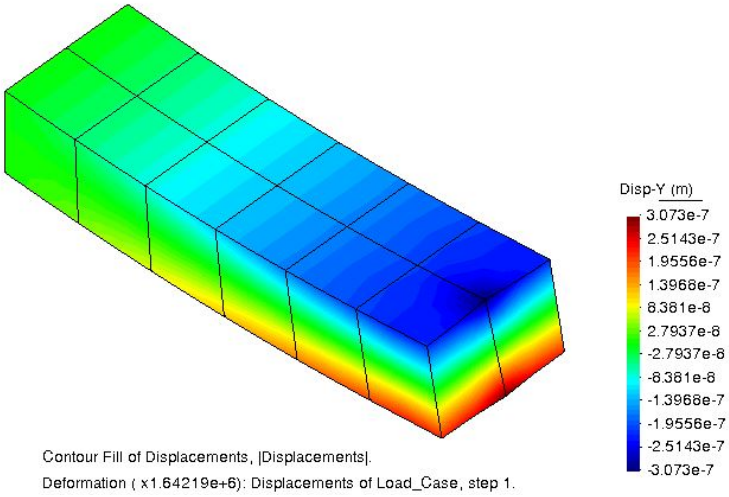


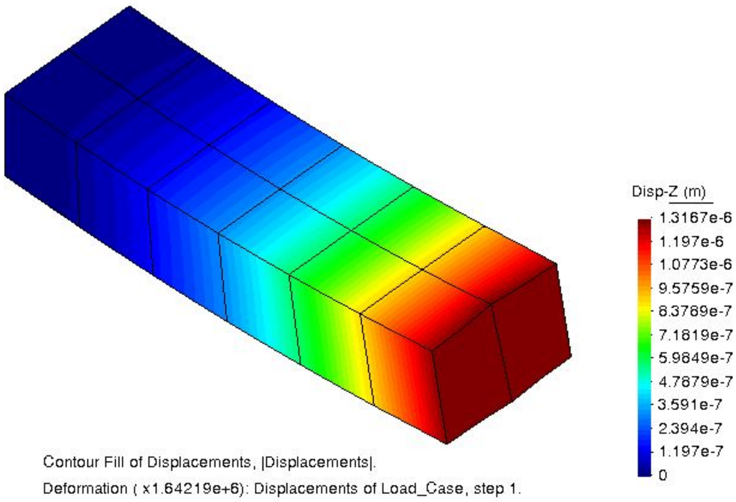
Figure 19: Deformation in the beam with maximum displacement nodal values

Next, the results obtained with 20 noded hexahedral elements are shown below:

<u>Case 2: 20 noded hexahedral - 12 elements and 127 nodes</u>



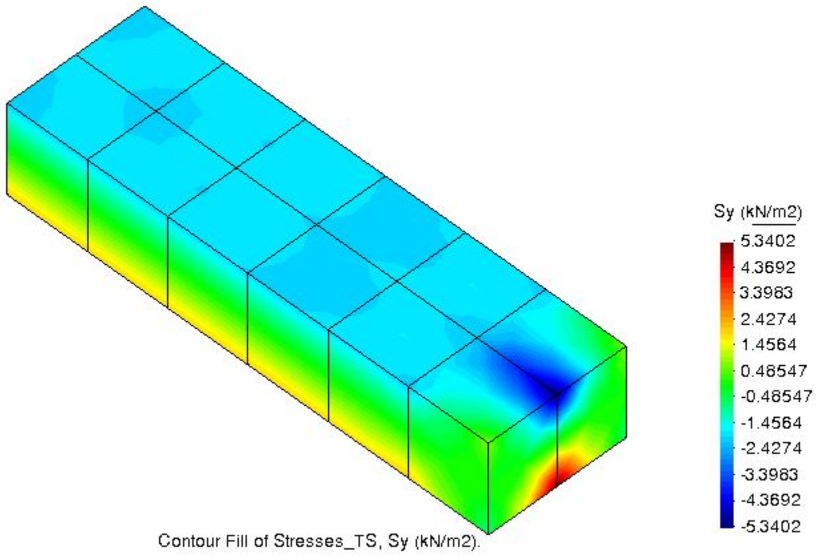
(a) y-displacement plot



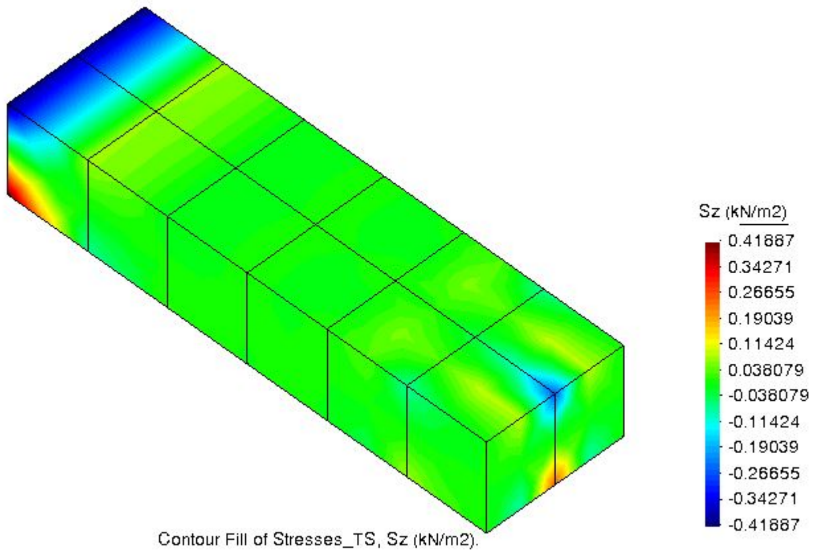
(b) z-displacement plot

Figure 20: Results: Displacement plot in the deformed configuration

The stress distribution plots obtained for this case are given next. Similar result patterns are obtained and it can be noticed that the gradient in the z-displacement causes the bending in the beam. To visualise the deformation in this case, Figure 22 shows the deformed configuration with the maximum z-displacement values.



(a) σ_y plot



(b) σ_z plot

Figure 21: Results: Stress distribution plot

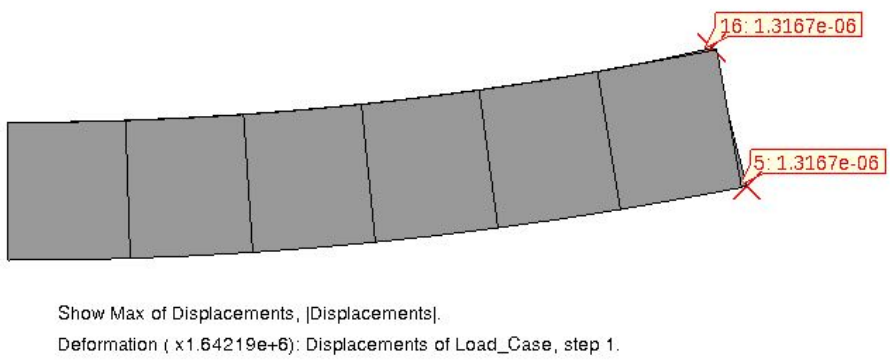


Figure 22: Deformation in the beam with maximum displacement nodal values

2.3. Comparison of the results

The results obtained are compared with the analytical solution and shown in Table 1. For calculating the analytical solution, we calculate the displacement of the free end of the cantilever beam where moment is applied. We know,

$$u_z^{analytical} = \frac{ML^2}{2EI} = \frac{12PhL^2}{2Ebh^3} = \frac{6PL^2}{Ebh^2} \tag{1}$$

here, the problem states that $P = 10000N$, $b = 6m$, $h = 4m$, $L = 21m$ and $E = 2.1\text{e}11 N/m^2$. Therefore we obtain,

$$\boxed{u_z^{analytical} = 1.3125e - 6\ m} \tag{2}$$

	No. of elements	No. of DoF	Max. u_z (m)	Error %
Hexahedral with 8 nodes	456	700	1.3042 e-6	0.63
Hexahedral with 20 nodes	12	127	1.3167 e-6	0.32

Table 1: Comparison between the numerical results obtained with different mesh elements

It is clearly seen that the solution is quadratic in nature and therefore using linear element is not a good choice to describe the behaviour of the beam. But with a finer mesh, the resultant error can be reduced to a given tolerance. Interestingly, with a very few elements (coarse mesh) compared to the linear case, the quadratic elements captures the behaviour of bending very effectively and is a better choice for this exercise.

Exercise 3: Foundation of a corner column

The figure shows a corner column with its foundation. This type of foundation is characterised by the fact that the support reactions are eccentric with respect to the load of the column. This results in a flexion of the column and lifting of the base slab. Analyse the state of stress in the column and the slab under the assumption that the slab is supported elastically by the ground. Determine whether or not the slab suffers lifting. Use hexahedrons with eight nodes.

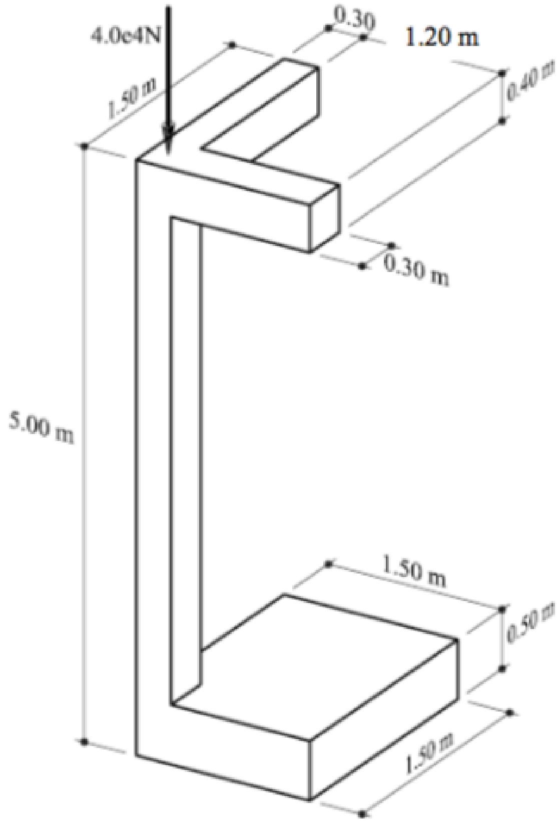


Figure 23: Foundation of a corner column

Solution:

3.1. Purpose of the exercise

The objective of this exercise is to analyse the behaviour of the structure and determine whether the slab suffers from lifting.

3.2. Analysis

3.2.1 Pre-processing

(i) Geometry

The first step of pre-processing is to model the geometry of the column and slab as per the given dimensions using extrusion in GiD shown in Figure 24.

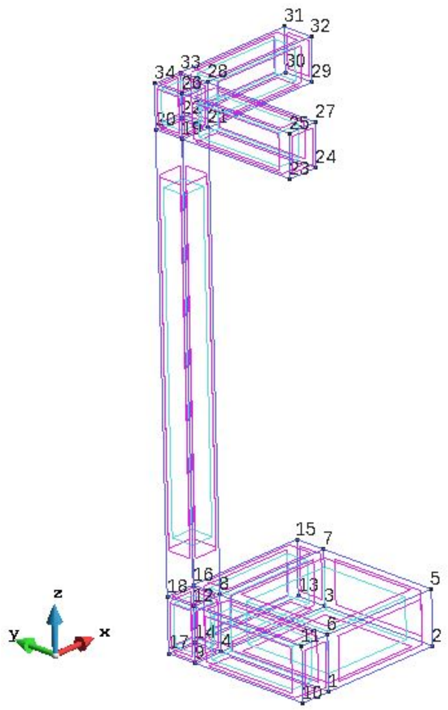


Figure 24: Defining the geometry

(ii) Data
Once the geometry is defined, we apply the given data to the model.

Problem type

As seen in the problem, we select the 3D Solids problem type from the Ramseries Educational module.

Boundary Conditions

Next, we define the boundary conditions for the problem as shown in Figure 25. For the given problem, the following conditions are set:

- Elastically supported ground is characterised by the ballast coefficient of the ground for all the bottom surfaces.
- Since it is assumed that the structure is connected to other symmetric structures at the top, a restriction on the X-Y plane translation is imposed at the end of the top beams to have zero displacement.

- The point load at the centre of the top surface of the column is modelled as a Global Projected Pressure over the surface.

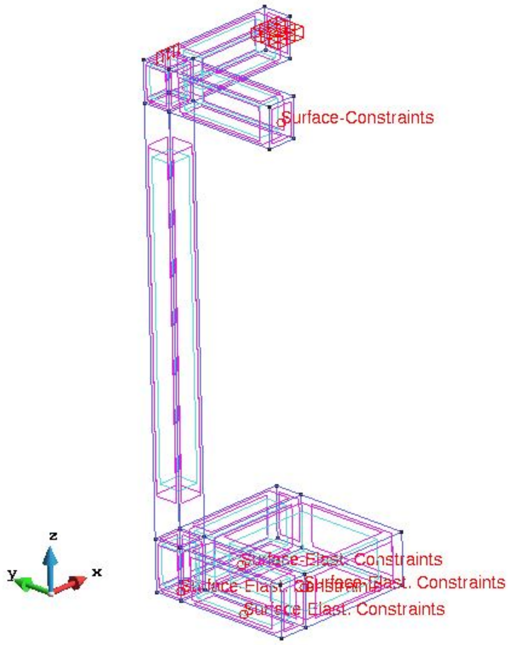


Figure 25: Boundary conditions

Material

The material properties of the concrete are defined as depicted in Figure 26.

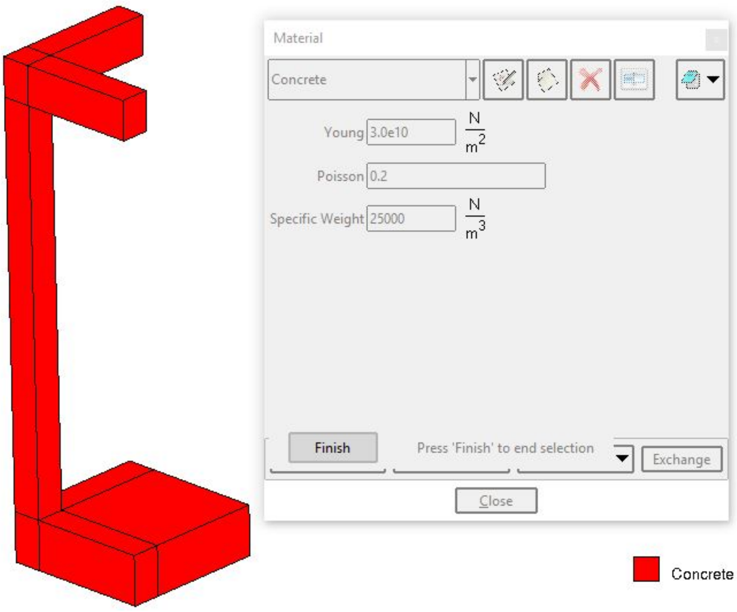


Figure 26: Material properties

Problem Data

In the problem data definition, we consider the self weight of the plate with a scale factor of 1.0. Figure 27 shown the necessary data used for this problem.

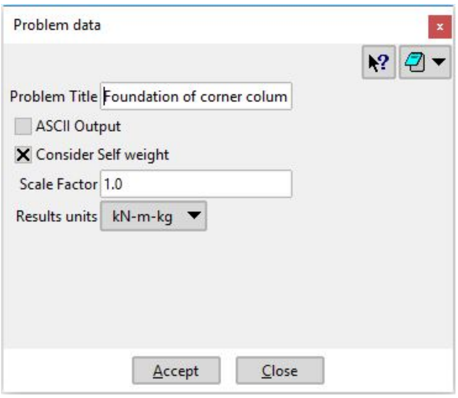


Figure 27: Problem data

Mesh

In this problem, a structured mesh is generated with linear hexahedron elements of $0.1m$ on its three dimensions. Figure 28 shows the structured mesh used with 8 noded hexahedron elements.

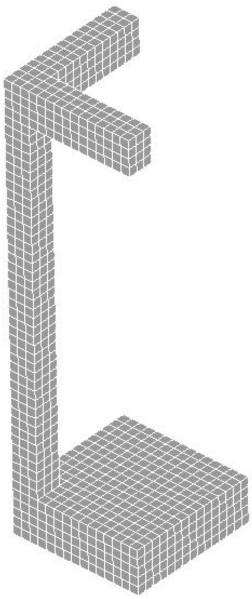


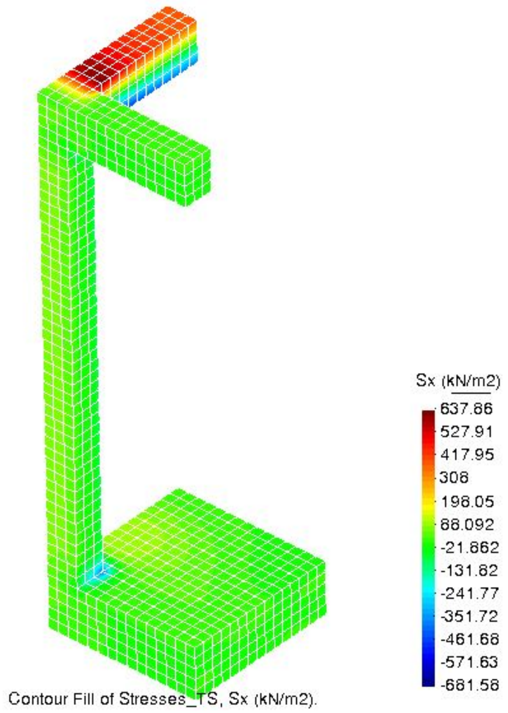
Figure 28: Structured mesh generation

3.2.2 Processing

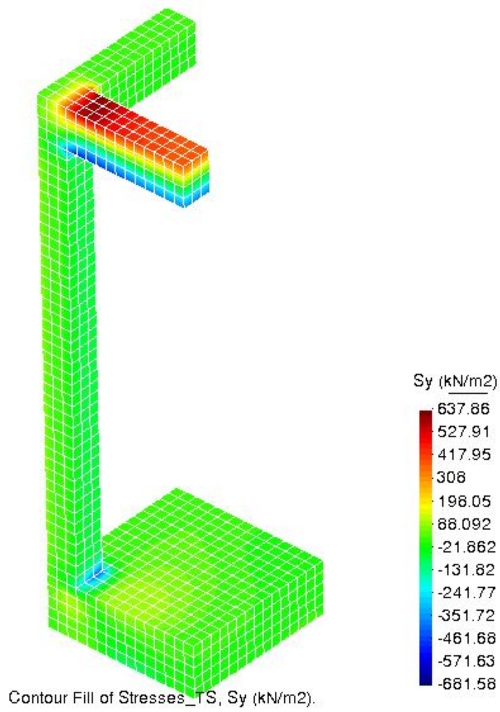
After all the data is defined and the mesh is generated, the simulation is run.

3.2.3 Post-processing

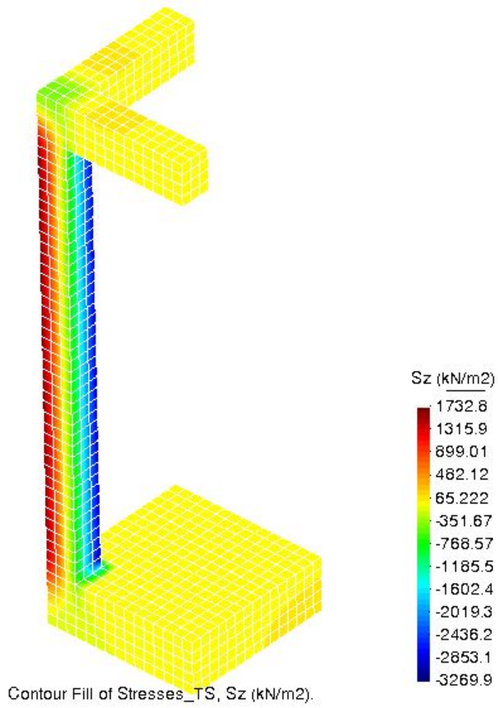
The following figures show the results obtained from the simulation. Firstly, the results obtained for the stresses in all directions are shown in Figure 29.



(a) S_x



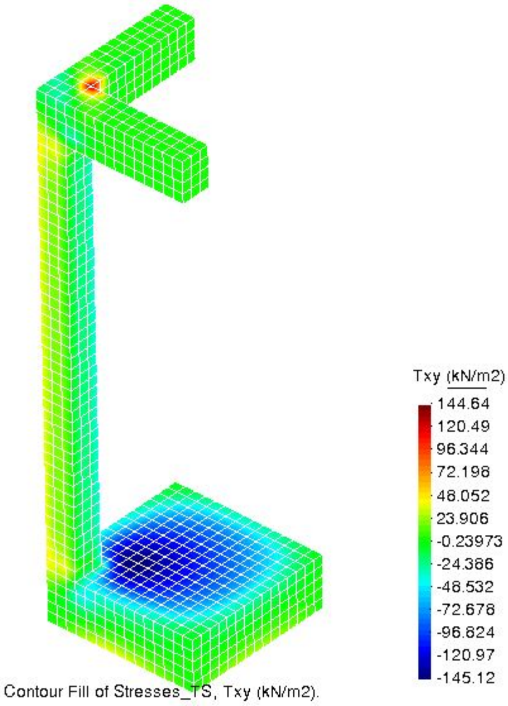
(b) S_y



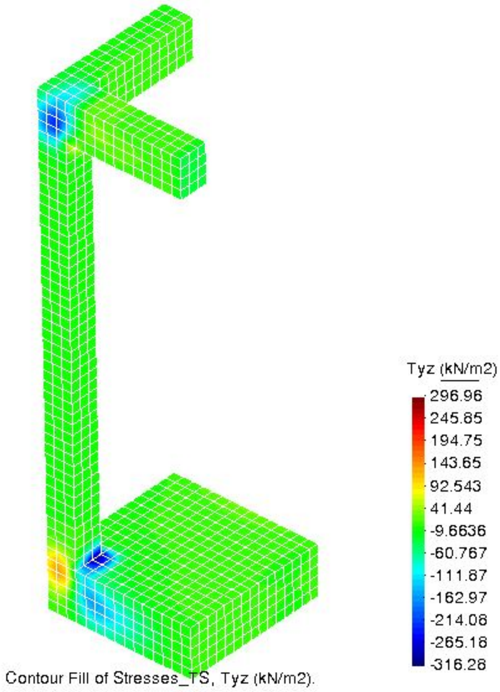
(c) S_z

Figure 29: Results: Stresses in the structure

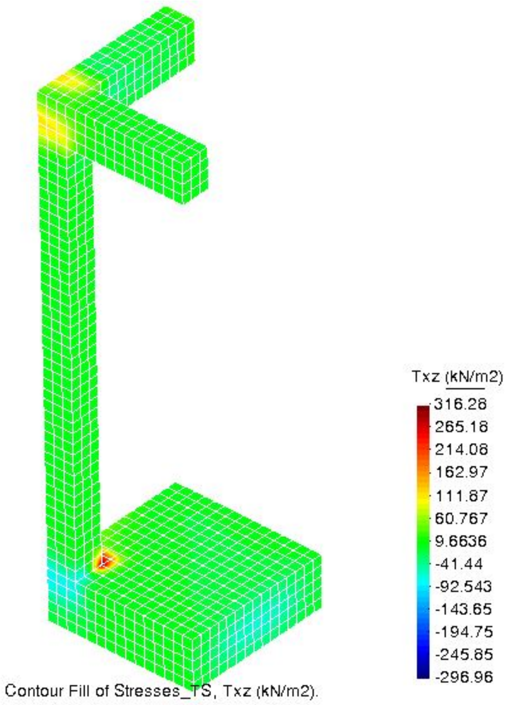
This is followed by the results obtained for shear stresses in the structure as shown in Figure 30.



(a) T_{xy}



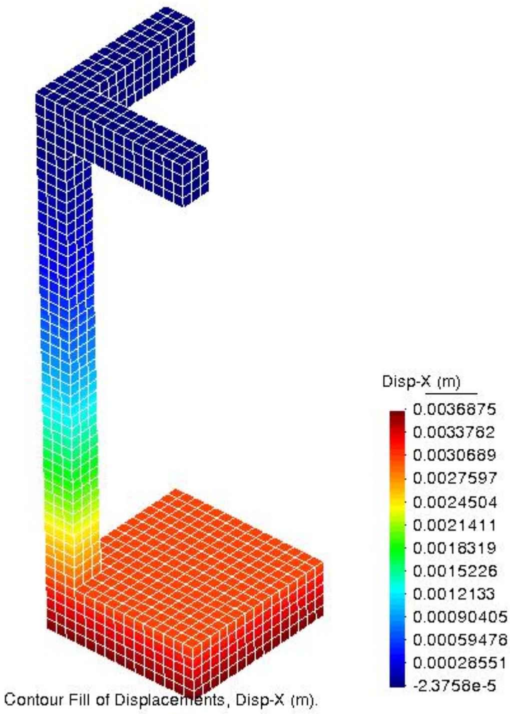
(b) T_{xz}



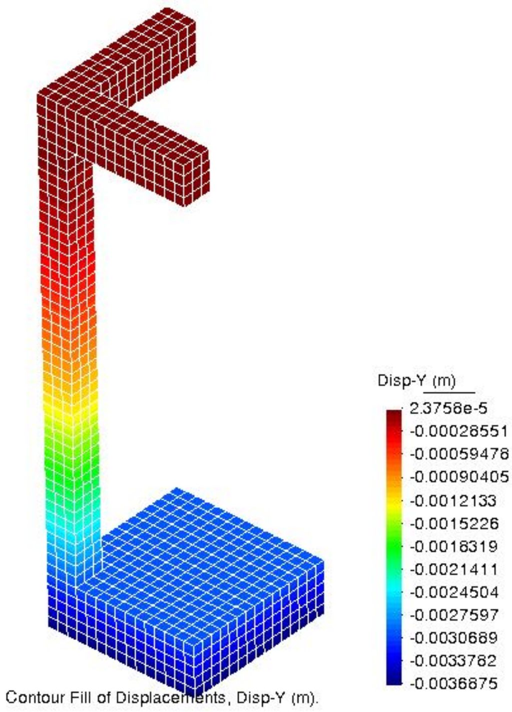
(c) T_{yz}

Figure 30: Results: Shear stresses in the structure

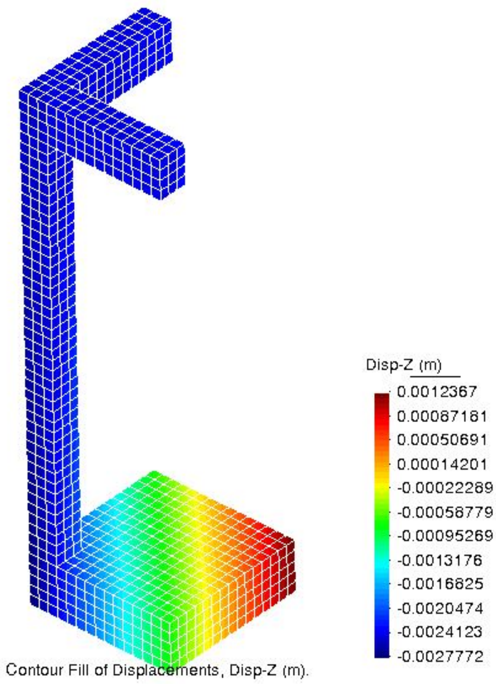
Next, the plots for the displacement of the foundation of a corner column in all directions is shown in Figure 31.



(a) $Disp - x$



(b) $Disp - y$



(c) $Disp - z$

Figure 31: Results: displacement in the structure

It is important to note that the displacement result in the z-direction allows us to determine whether the slab suffers from lifting or not. For this, the displacement in the z-direction is plotted on deformed configuration and shown in Figure 32.

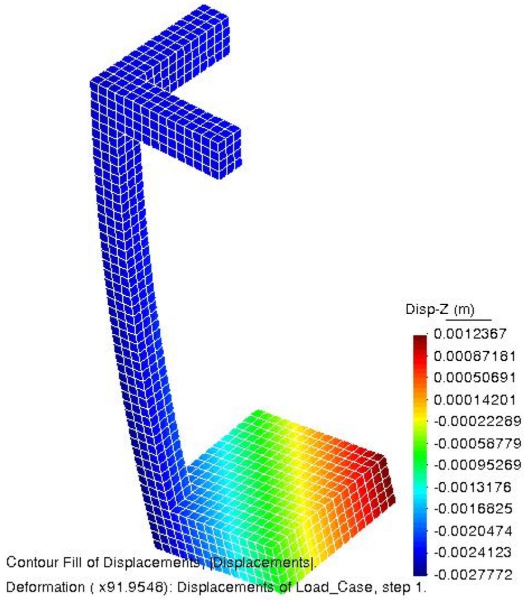


Figure 32: Displacement in the z-direction plotted on deformed configuration

3.3. Comparison of the results

Firstly, the distribution of stresses over the structure given in the results shows:

- identical stresses in x and y directions due to symmetry.
- maximum stress values concentrated on the top bars due to the displacement restriction imposed and the consequent reaction forces.
- equal and opposite values obtained for top and bottom part gives a clear sign of moment.
- large shear stress value T_{xy} due to the elastic constraint imposed on the bottom surface due to soil.

The results obtained for the displacement provides:

- symmetric results in x and y-directions.
- small but not negligible displacements in x and y-directions.
- displacements in z-direction produces downward displacement at one end and upward displacement at the other end of the foot.

Considering the deformed configuration shown in Figure 32, we see that a part of the structure's base is lifted. It is important to understand that in case of higher loading, the lift condition would only increase and proper measures should be taken to curb this potential of failure. Providing a bigger size footing could distribute the stresses and increase the weight of the structure to counter lift making it a safer structure.