Computatioanl Structural Mechanics and Dynamics

Homework 1

Juan Diego Iberico Leonardo

Zahra Rajestari

Oriol Call Pinol

The following document is presented in the framework of structural analysis for the given problems. All the process is done using the software GID and the extension RAMSERIES EDUCATIONAL 2D for plane state problems.

In the first problem, a developed convergence analysis was done comparing different mesh elements of different orders. In problem 2, the structure of a two frame bay with circular holes is analyzed, where the imposition of a displacement on vertical direction plays an important role in the evaluation of the frame.

In problem 3, a concrete beam with a hole for ducts facilities is studied. The importance of this problem arises from two steel plate reinforcement that is placed around the hole.

The last problem deals with a geometry refer to a dam with linear loads and the imposition of specific boundary conditions for the soil.

Problem 1

1. Geometry and boundary conditions

The geometry of the structure is defined and boundary conditions and loads are imposed.

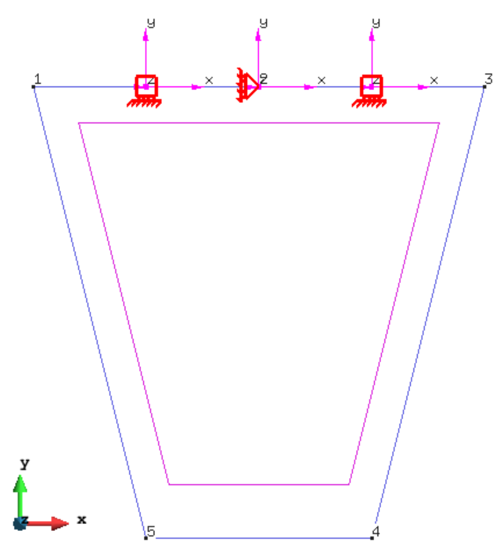


Figure 1. Geometry of the structure

2. Problem data

Problem type for this structure is chosen to be *“Plane state”* and since the structure is under its self-weight, this option should be activated in problem data:

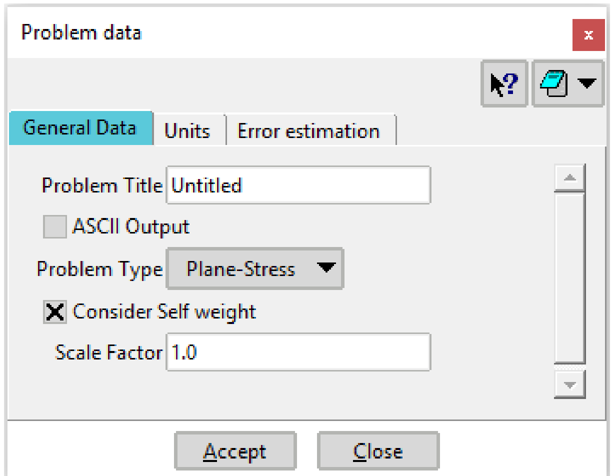


Figure 2. Problem data

3. Mesh

The structure is to be analyzed using triangular elements with 3 and 6 nodes and quadrilateral element with 4, 8 and 9 nodes. The mesh for each case is shown in the following:

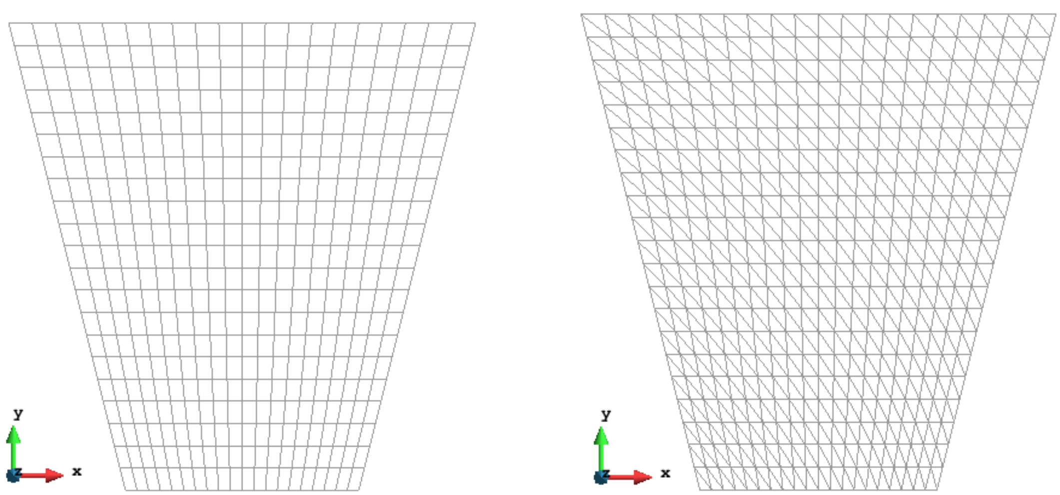


Figure 3. Mesh of the structure with quadrilateral and triangular

4. Results and discussion

The stress and the displacement distribution in y-direction are shown for each element type using different number of nodes. In general, for a specific element type, as the number of nodes increase the results show better accuracy. Therefore, triangular element with 6 nodes and quadrilateral element with 9 nodes offer better results and among these two, quadrilateral element with 9 nodes has better accuracy in comparison to the exact solution.

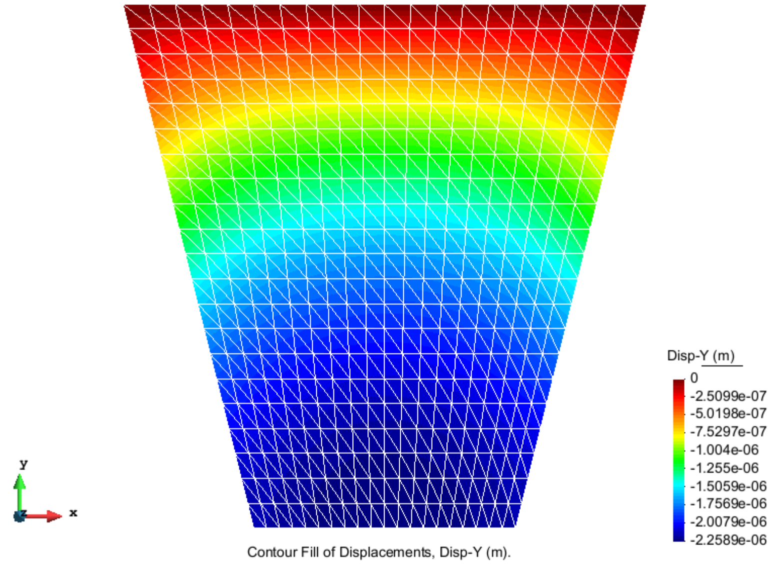


Figure 4 Displacement in Y direction for triangular elements with 3 Nodes

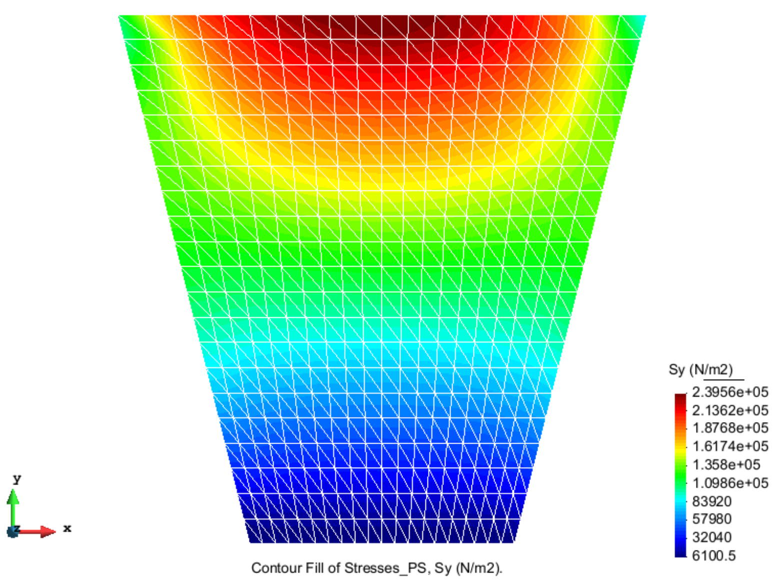


Figure 5 Stress in Y direction for triangular elements with 3 Nodes

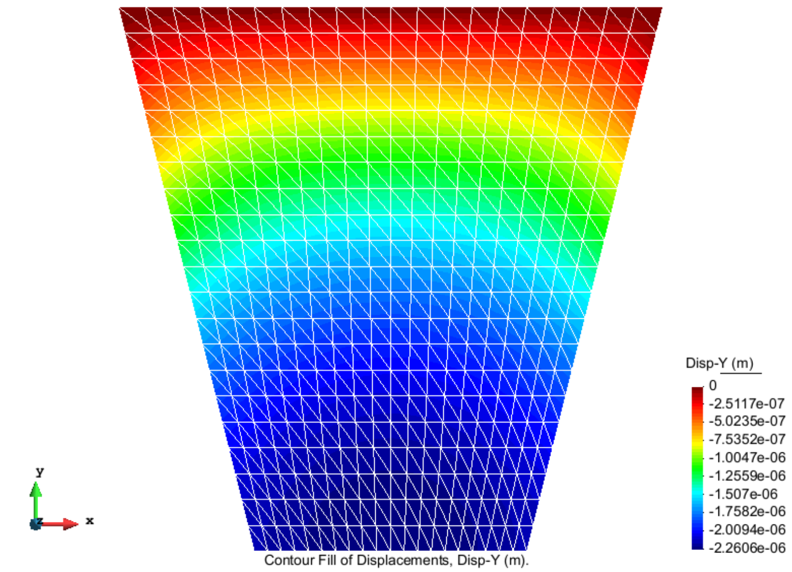


Figure 6 Displacement in Y direction for triangular elements with 6 Nodes

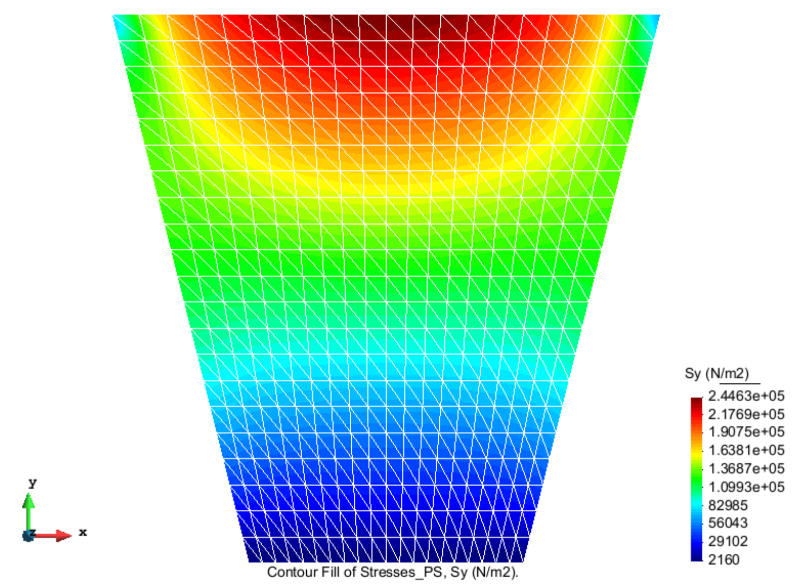


Figure 7 Stress in Y direction for triangular elements with 6 Nodes

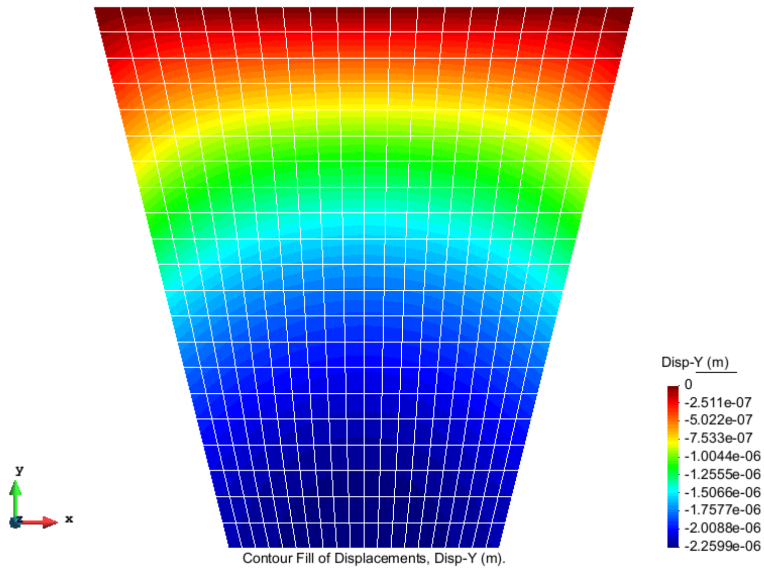


Figure 8 Displacement in Y direction for rectangular elements with 4 Nodes

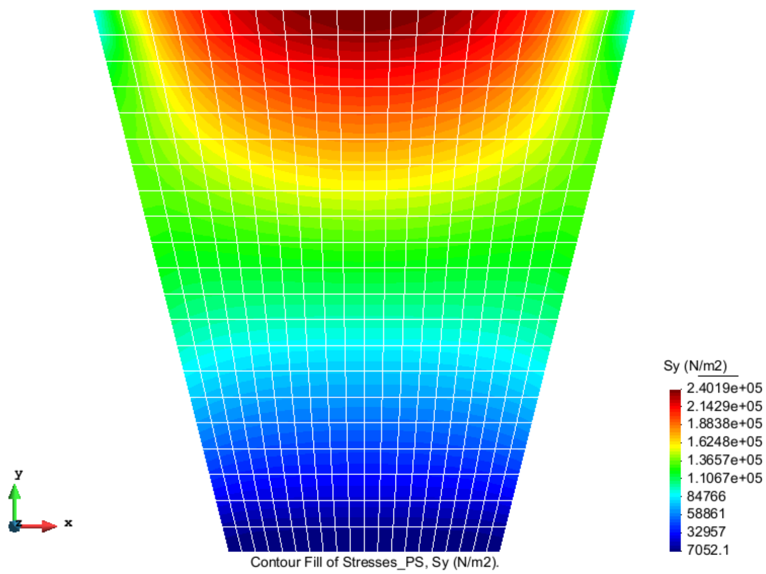


Figure 9 Stress in Y direction for rectangular elements with 4 Nodes

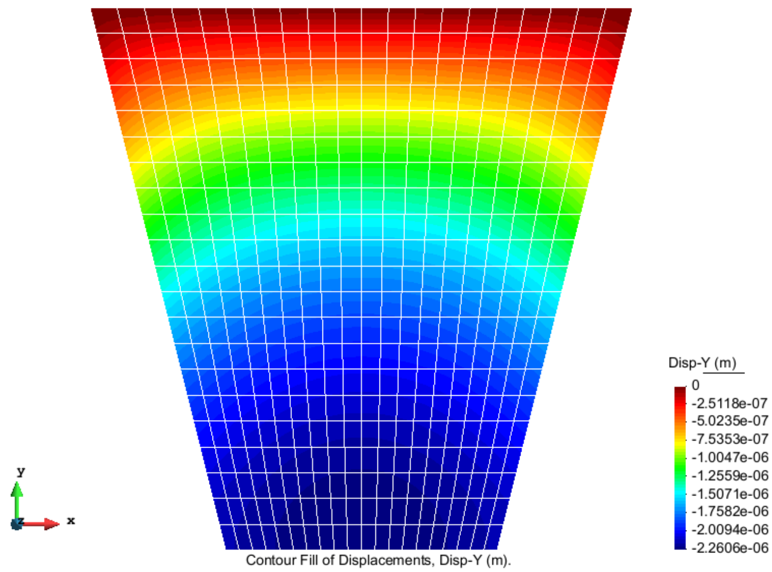


Figure 10 Displacement in Y direction for rectangular elements with 8 Nodes

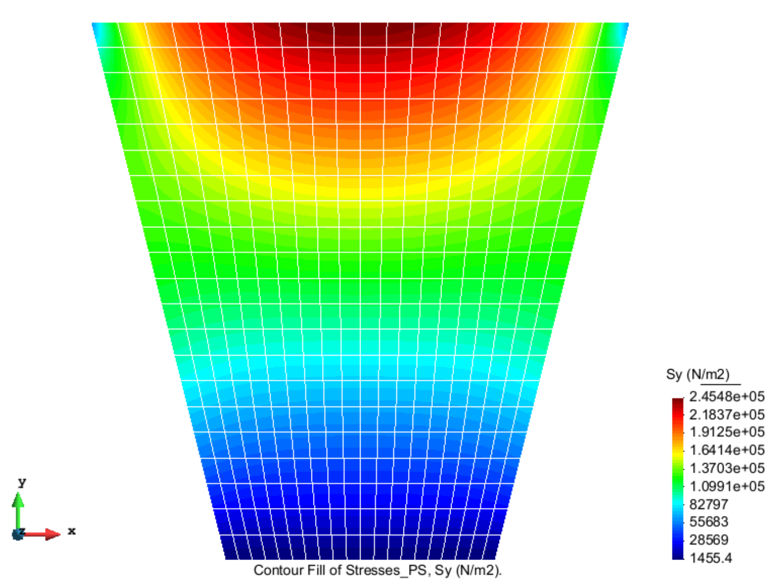


Figure 11 Stress in Y direction for rectangular elements with 8 Nodes

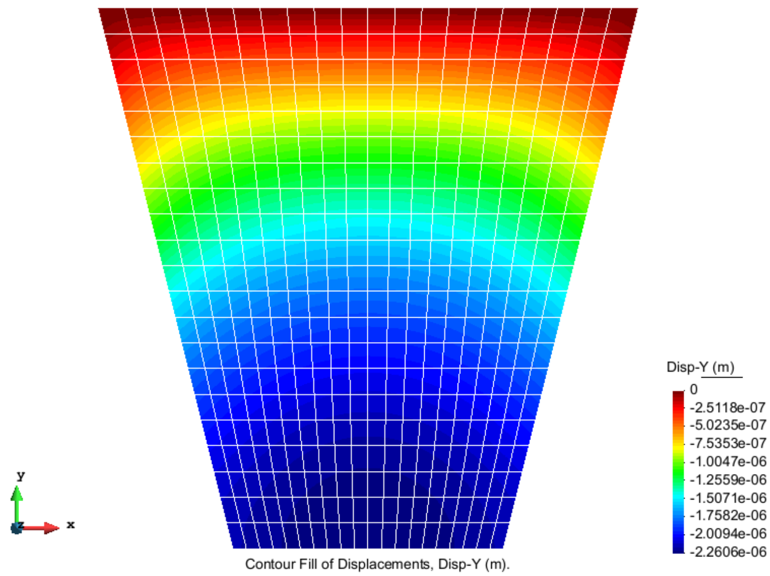


Figure 12 Displacement in Y direction for rectangular elements with 9 Nodes

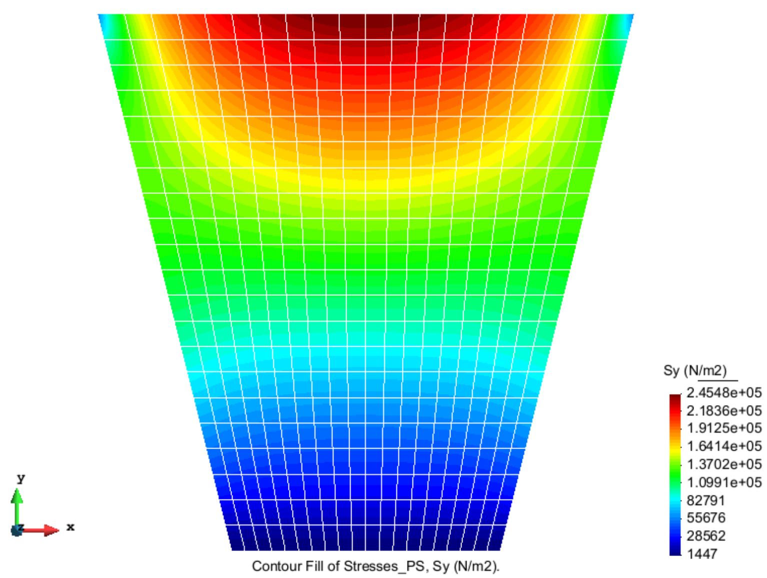
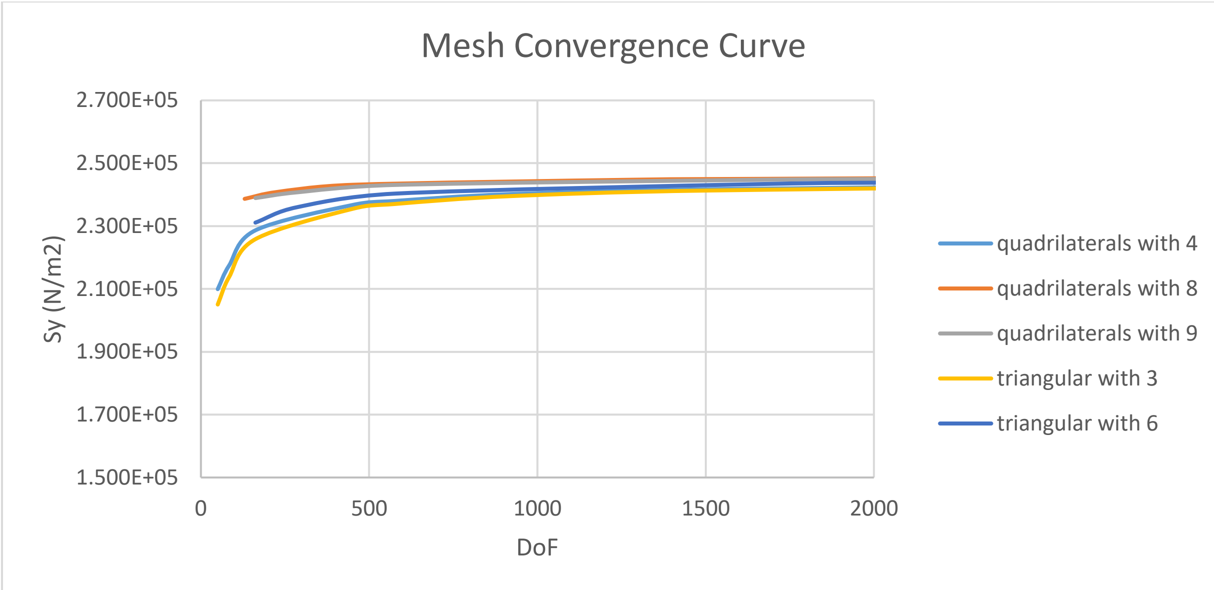


Figure 13 Stress in Y direction for rectangular elements with 9 Nodes

The structure is analyzed and results for stress and displacement are observed for different numbers of DoFs (degrees of freedom) by changing the size of the mesh. The convergence graph is plotted afterwards in order to find suitable mesh for the analysis. In the following table, the errors of each case is computed with respect to the exact solution.

	DOF	max sy(N/m2)	max disp(m)	%error Sy	%error disp.
triangular with 3	50	2.050E+05	-2.28600E-06	16.98583	1.15E+00
	84	2.138E+05	-2.26867E-06	13.42591	3.84E-01
	162	2.258E+05	-2.25845E-06	8.565182	6.86E-02
	450	2.355E+05	-2.25814E-06	4.663158	8.23E-02
	578	2.371E+05	-2.25847E-06	4.027126	6.77E-02
	924	2.396E+05	-2.25892E-06	3.012955	4.78E-02
triangular with 6	162	2.311E+05	-2.25654E-06	6.434008	1.53E-01
	286	2.360E+05	-2.25850E-06	4.44413	6.64E-02
	578	2.403E+05	-2.25951E-06	2.692713	2.17E-02
	1682	2.434E+05	-2.25986E-06	1.453441	6.19E-03
	2178	2.439E+05	-2.26012E-06	1.261943	5.31E-03
	3526	2.446E+05	-2.26057E-06	0.95749	2.52E-02
quadrilaterals with 4	50	2.099E+05	-2.26589E-06	15.00567	2.61E-01
	84	2.175E+05	-2.26375E-06	11.94372	1.66E-01
	162	2.287E+05	-2.26046E-06	7.427935	2.04E-02
	450	2.367E+05	-2.25998E-06	4.171255	8.85E-04
	578	2.380E+05	-2.25995E-06	3.636437	2.21E-03
	924	2.402E+05	-2.25990E-06	2.75587	4.42E-03
quadrilaterals with 8	130	2.387E+05	-2.25841E-06	3.37247	7.04E-02
	226	2.409E+05	-2.25925E-06	2.489879	3.32E-02
	450	2.431E+05	-2.25975E-06	1.563158	1.11E-02
	1290	2.448E+05	-2.25990E-06	0.906883	4.42E-03
	1666	2.450E+05	-2.26014E-06	0.796761	6.19E-03
	2686	2.455E+05	-2.26058E-06	0.61498	2.57E-02
quadrilaterals with 9	162	2.389E+05	-2.25869E-06	3.268826	5.80E-02
	286	2.408E+05	-2.25939E-06	2.52753	2.70E-02
	578	2.431E+05	-2.25979E-06	1.576113	9.29E-03
	1682	2.448E+05	-2.25990E-06	0.909717	4.42E-03
	2178	2.450E+05	-2.26015E-06	0.798785	6.64E-03
	3526	2.455E+05	-2.26059E-06	0.615789	2.61E-02

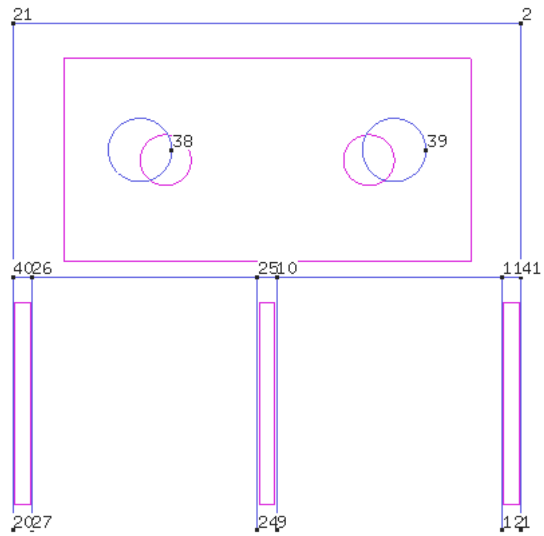


According to the graph provided, it can be seen that using 1000 DoFs, which corresponds to a mesh size of almost 0.2 for all element types, offers good convergence with quadrilateral elements of order 2 with nine nodes.

Problem 2

1. **Geometry**

   The geometry of the structure is defined as the following:

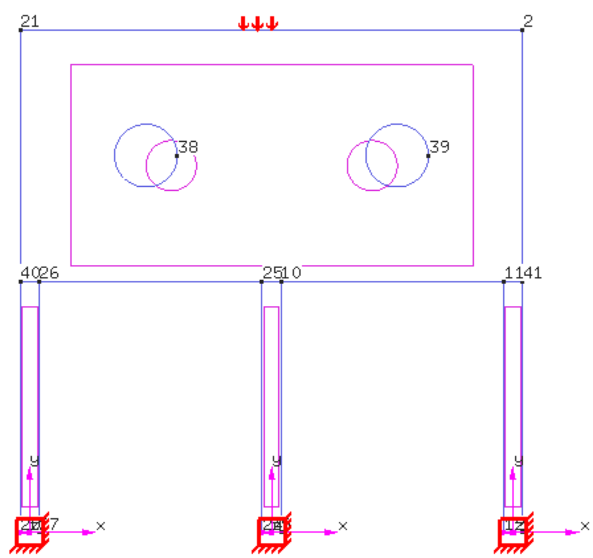


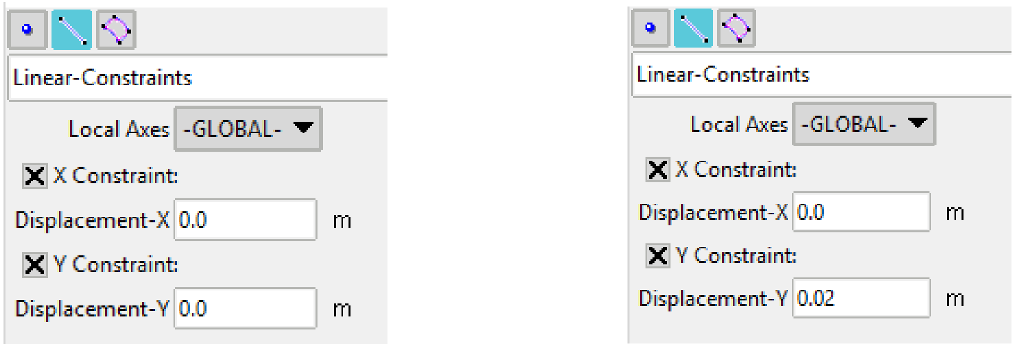
Geometry of the structure

2. **Problem data, Boundary conditions and loads**

Problem type for this structure is defined as *"plane state"*. The problem is assumed to have the hypothesis of plane stress which should be defined in *"problem data"*.

This problem is solved in two different cases. The first case is assuming no displacement for any of the columns, and the second case is imposing a displacement equal to some arbitrary values to the second column. The loading and boundary conditions are applied accordingly and can be found in the following:





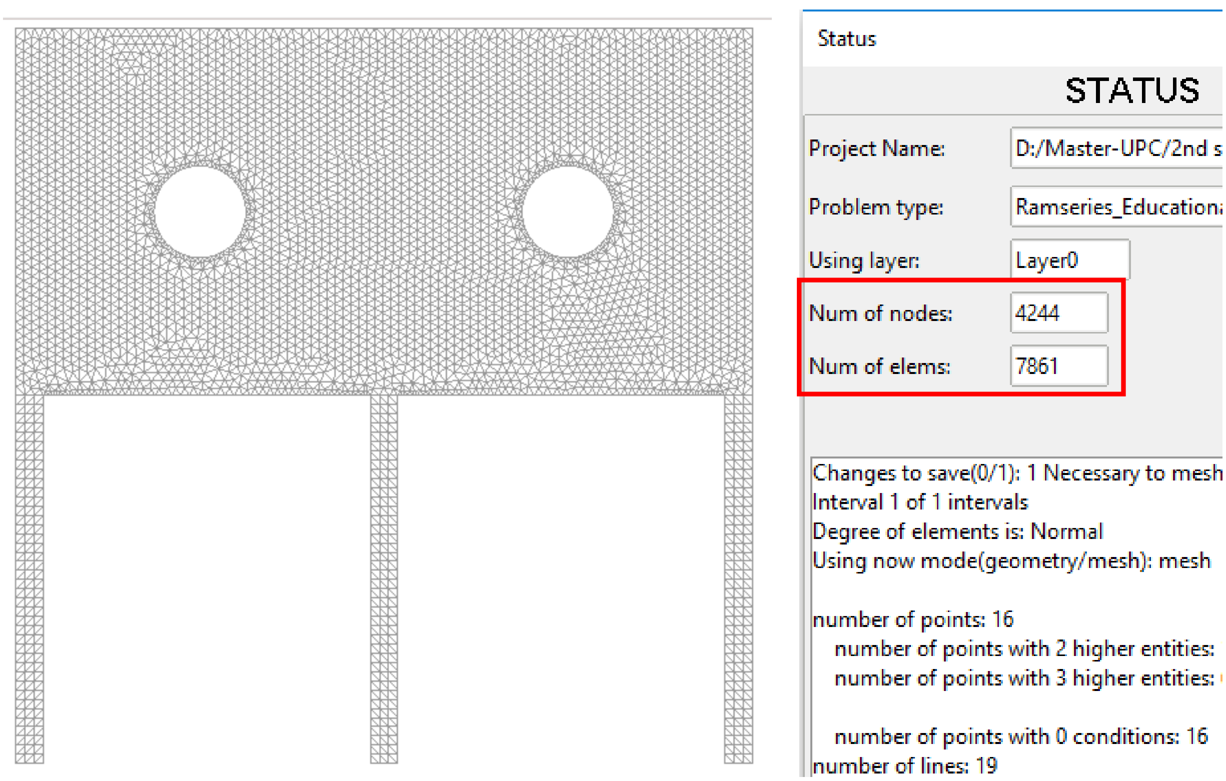
a) $\delta = 0$ b) $\delta = 20\ cm$

Loading and boundary conditions of the problem

The value for the uniform distributed force and the material properties are assigned based on what is defined for the problem.

3. Mesh

Triangular elements with 3 nodes are used to mesh the structure. In order to have a better mesh, the structure is divided into 4 surfaces three of which are the columns holding the plate and the other one is the plate itself. The mesh setting is assigned as structured mesh on surface with cell size of 0.1 and the mesh in the corners of the structure and the region around the circles is refined using point and line mesh with size of 0.05. The mesh can be found in the following figure:

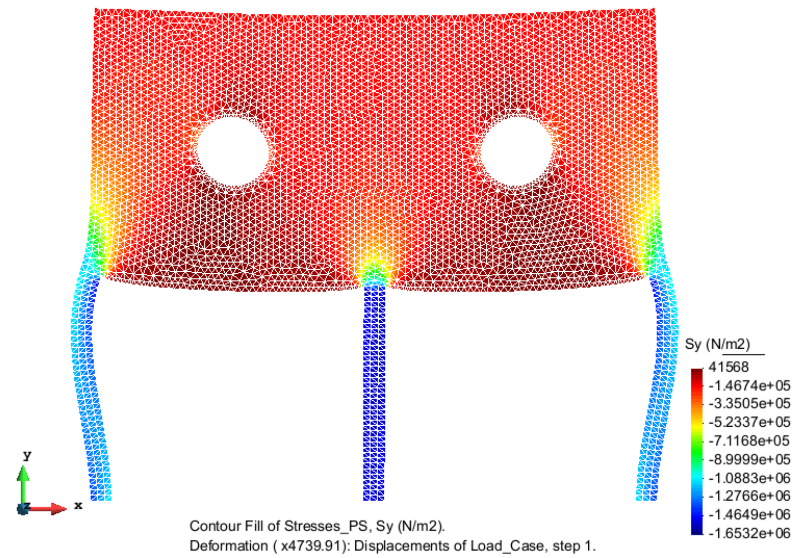


Triangular mesh for the structure

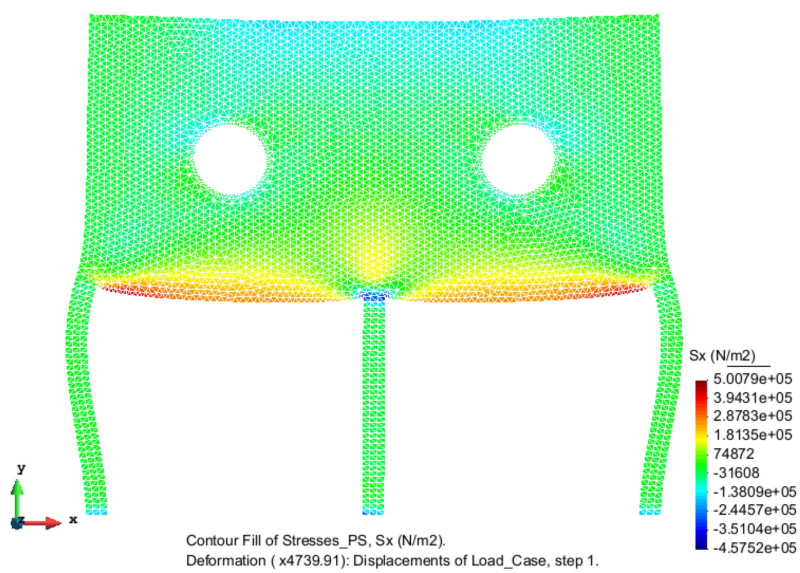
4. Results and discussion

Now that the mesh is generated and all the other conditions are imposed to the structure, the calculation process starts. The stress distribution is shown below for the two cases explained before.

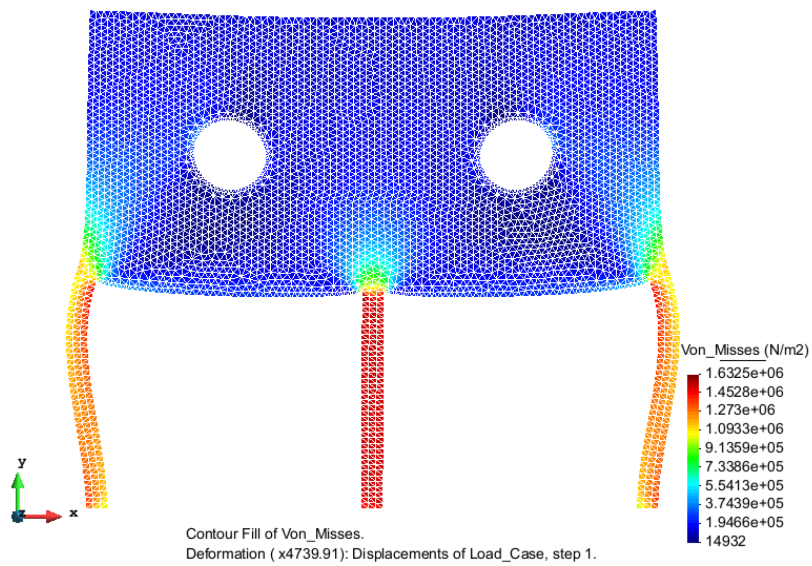
The stress distribution is shown for stress in x-direction, y-direction and von-mises stress.



The stress distribution in y-direction shows that the structure is experiencing high compression in the columns. It can also be seen that according to the load path the stress under the holes is much less than that compared to other parts of the structure.

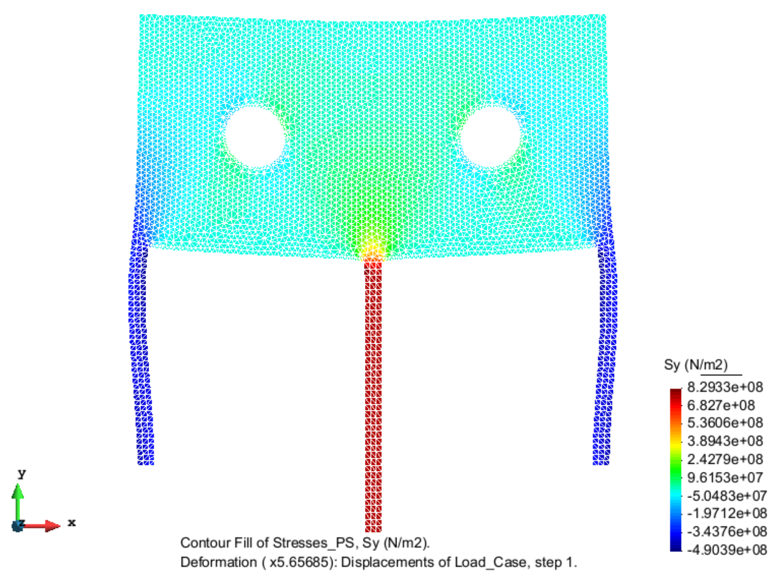


Since the load is applied in y-direction and also the displacement to the structure is imposed downwards, the stresses in x-direction are not critical. Only in the down edges of the plate the stress in x-direction is high which is because of the bending moment created from the load. And also in the critical parts of the geometry that is in corners the structure is undergoing high stress.

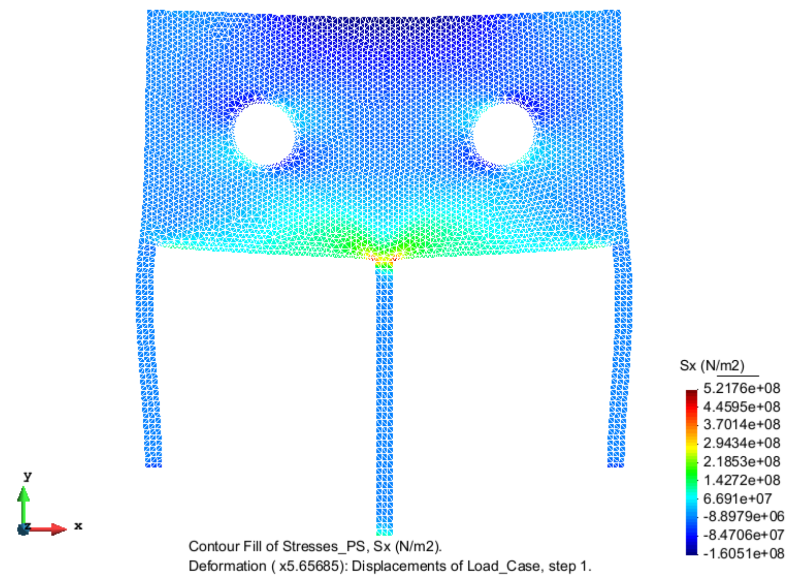


The von-Misses stress distribution shows that the second column have the highest stress and it is the critical part of the structure.

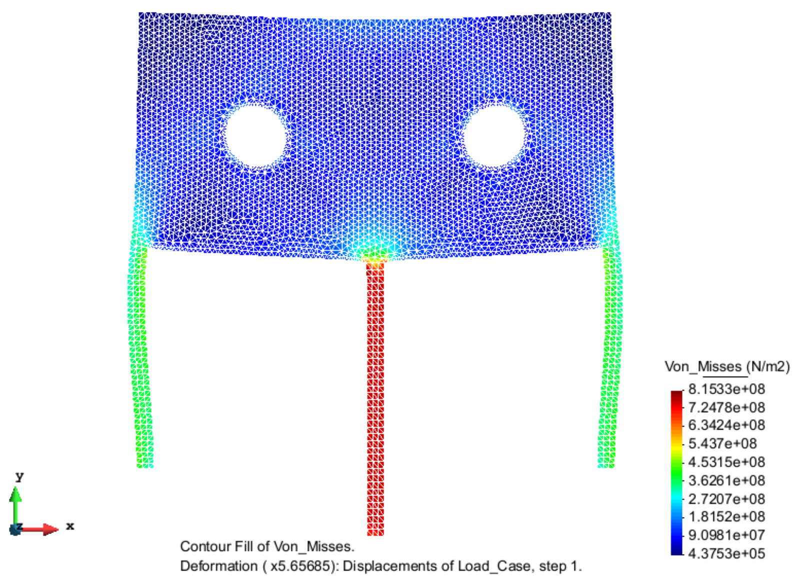
The results for case of imposing a displacement of 20 cm is shown below:



According to the figure for stress in y-direction, the middle column has a high tensile stress due to the displacement imposed in y-direction. All the compression is taken by the other columns creating a bulb of forces on the beams.



Based on the figure shown for stress in x-direction, we can say that compared to the previous case the stresses are higher. But in this case the highest tensile stress is focused on the transition parts from plate to the mid column.



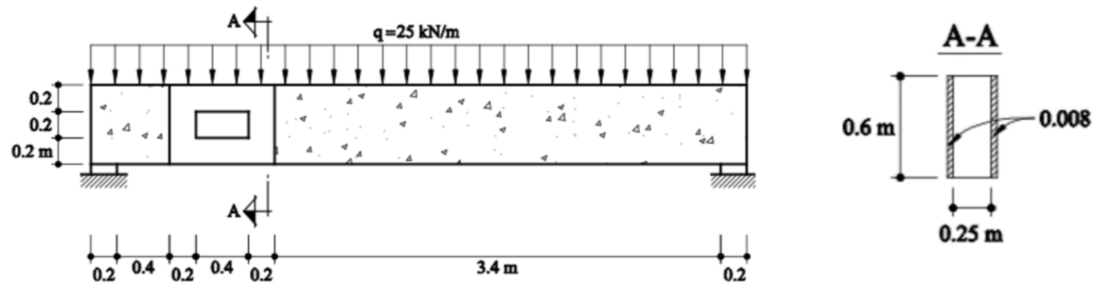
Von-Misses distribution shows that as expected the middle column which is tolerating the highest tension.

Problem 3

1. Description:

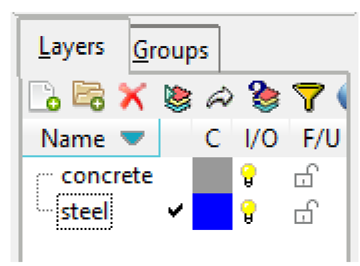
The following practice was developed in the frame of plane-stress using the software program GID to represent the geometry and their peculiarities. In order to calculate the structure, it was used the extension RAMSERIES EDUCATIONAL 2D.

The problem to be analyzed, is a structural type. The main element is a concrete beam with simple supports of 5 m of span, 0,6 m of height and 0,25 cm of wide. Due to the nature of the problem, rise the need to create a hole for facilities ducts. In order to reinforce the beam, two steel plates are placed on the area of the hole in order to provide more stiffness and consequently increase the maximum carrying load.

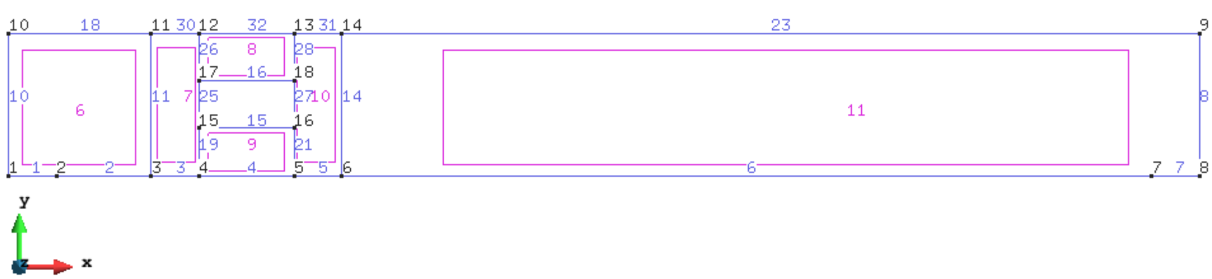


2. Geometry:

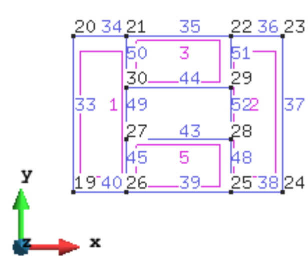
The geometry definition had been made in two different layers, one specific for the beam and another one for the steel plates. The separation by layers allows us to assign and model the elements with different properties.



The beam is composed by 18 nodes and 6 surfaces. The division in different surfaces, as it will be seen it, is giving more accurate mesh location in order to collapse the mesh nodes between the beam and the steel plate.



On the other hand, the second layer correspondent with the steel plate is formed by 11 nodes and four surfaces. That element must match perfectly with the beam geometry.



3. ***Boundary conditions:***

Respect the boundary conditions, all of them are applied on the concrete element. The supports (bearing the beam 0,2 cm in each extreme) are defined in a such way that the x and y displacement are restricted.

The load definition is imposed by a distributed load of value -25 kN/m on the top of the beam.

4. **Material properties:**

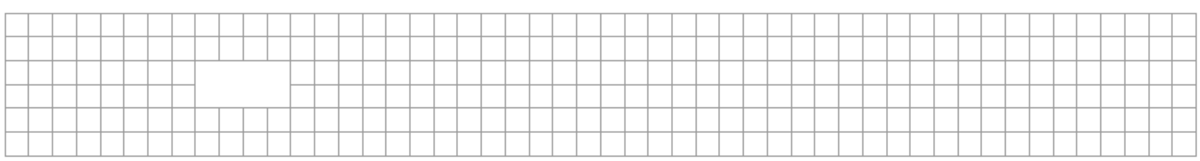
The material is given by the exercise,

$$\text{Concrete}\begin{cases} E = 3.0e10\,\dfrac{N}{m^2} \\ \nu = 0.2 \\ t = 0.25\text{ m} \end{cases} \qquad \text{Steel}\begin{cases} E = 2.1e11\,\dfrac{N}{m^2} \\ \nu = 0.3 \\ t = 0.016\text{ m (Two sheets of 0.008m)} \end{cases}$$

It can be seen, that both material properties are applied on each element separated by layers, defining the two plates thickness 0.008 m x 2 = 0.0016 m.

5. ***Mesh:***

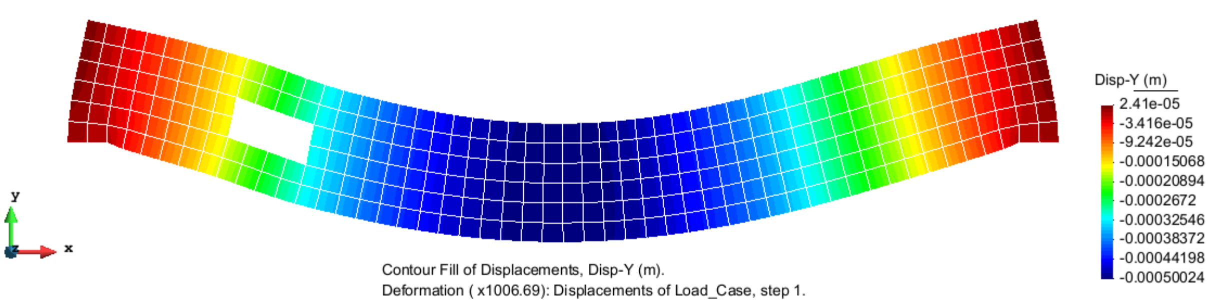
The mesh used for the elements is *structure mesh* of quadrilateral linear elements of 0.1 m size for simplicity. The key point in this stage is match the mesh nodes of both regions in common (beam and steel plate) in order to collapse the nodes and allow to behave as the real composed beam.



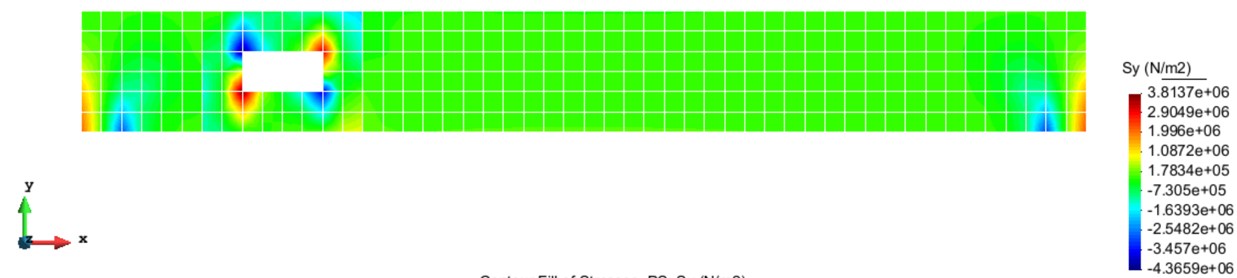
6. **Post process:**

Once solved the structural problem, RAMSERIES EDUCATIONAL 2D provide the results that will be analyzed:

<u>Deformation:</u> as it is expected, the maximum deflection it takes place on the mid-span, where we find the maximum value of positive bending moment. The 0,2 m of bearing length remain fixed and arising the maximum negative bending moment.

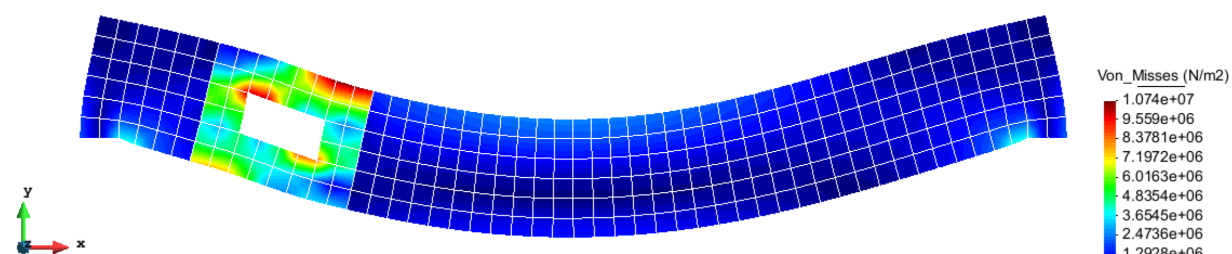


Stress: The stress on y direction it is shown in the following figure. We can appreciate how the steel plate is taking the shear force, giving the maximum stress on tension and compression. On the other side, on the bearing supports arise tensile stress as result for the *y* reaction and compression for the *x* restriction.



Contour Fill of Stresses_PS, Sy (N/m2).

Von Misses: The main stress values are located on the hole area, where it arises the most problematic region from the point of view of shear. On the other side, we find the usual compression areas on the top of the beam and tensile stress on the bottom as in this case on the reactions.



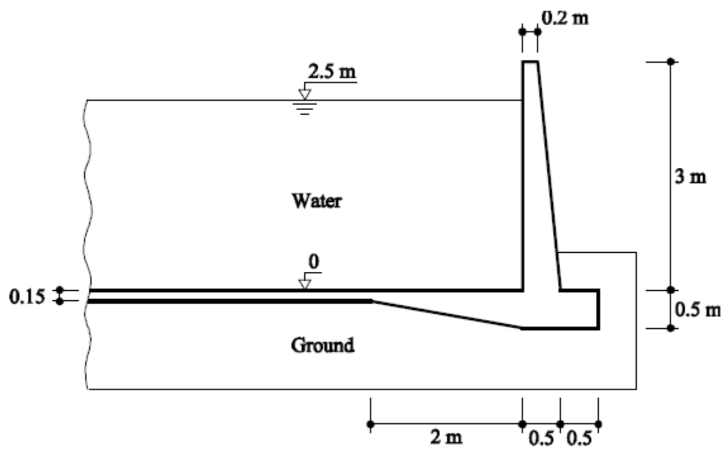
Contour Fill of Von_Misses.
Deformation (x1006.69): Displacements of Load_Case, step 1.

Problem 4

1. Description:

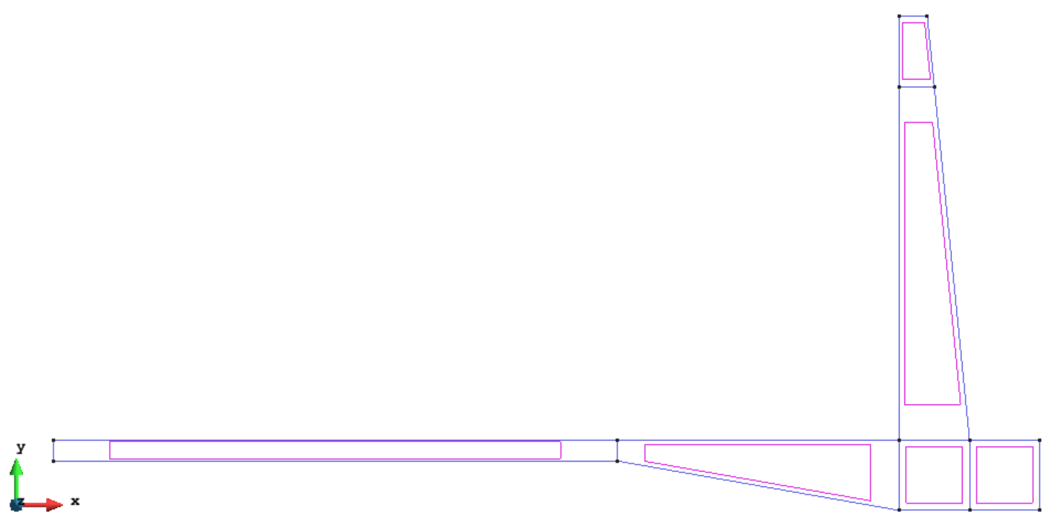
The following practice was developed in the frame of plane-stress using the software program GID to represent the geometry and their peculiarities. In order to calculate the structure, it was used the extension RAMSERIES EDUCATIONAL 2D.

The problem to be analyzed, is a structural type. The main element is a concrete dam with thickness 0.1m and located on the ground with elastic properties and containing water. Due the nature of the system we were supposed to assume symmetries in the left part of the problem.



2. Geometry:

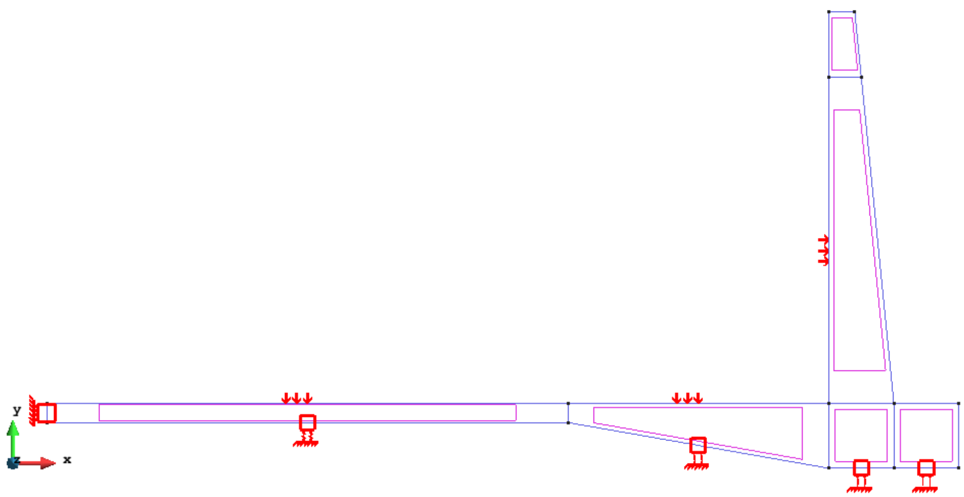
The geometry has been done in the most simplest way: drawn by lines and then divided in simple quadrilaterals to make the meshing a bit simpler.



3. ***Boundary Conditions:***

For this problem we established elastic constraints in the Y direction for the ground as we can see in the figure above and a displacement constraint on the X direction in the left part of the base. \

For the forces, we applied a distributed load on the base of the dam and a linear distributed load on the interior part of the wall. This is due to the pressure the water exert on the wall depends on the height such as: ρgh.



4. ***Material Properties:***

The material we applied to the system is concrete, given by the problem, the ground is not represented as a main material but just represented as an elastic constraint.

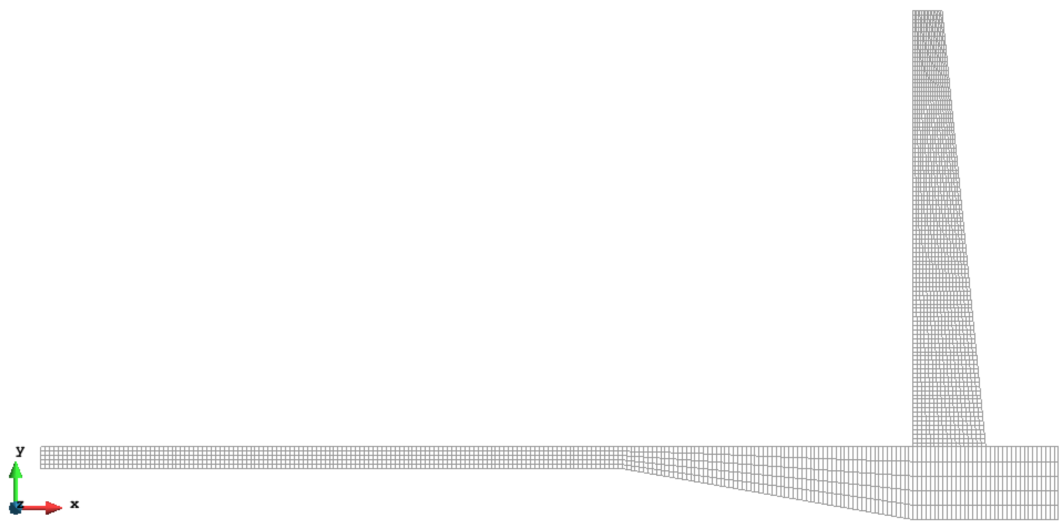
Data

$$\text{Concrete}\begin{cases} E = 3.0e10\,\dfrac{N}{m^2} \\ \nu = 0.2 \\ \gamma = 24000\,\dfrac{N}{m^3} \end{cases}$$

$$\text{Ground}\left\{\text{Load coefficient} = 50\,\dfrac{N}{cm^3}\right.$$

5. ***Mesh:***

The mesh used for this problem is composed by 0.1 Quad4 elements and it was applied to the system using Structured Surfaces from GiD software.

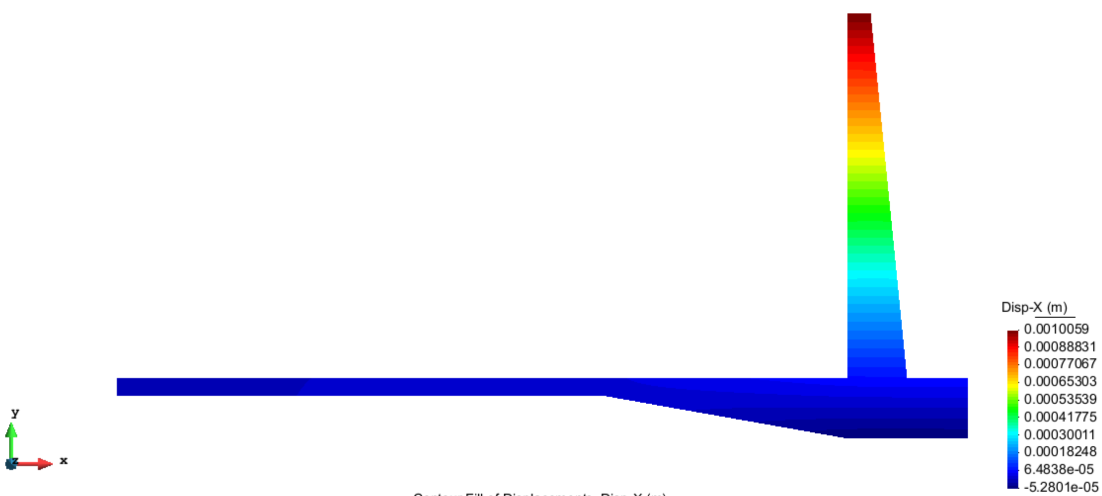


6. ***Postprocess:***

The last part, just after creating the mesh and calculating, is analyzing the results que get. The displacements and stresses.

<u>Displacements:</u>

The displacements in X show that the upper right part is the one that moves the most due to its far location from where the forces are applied.



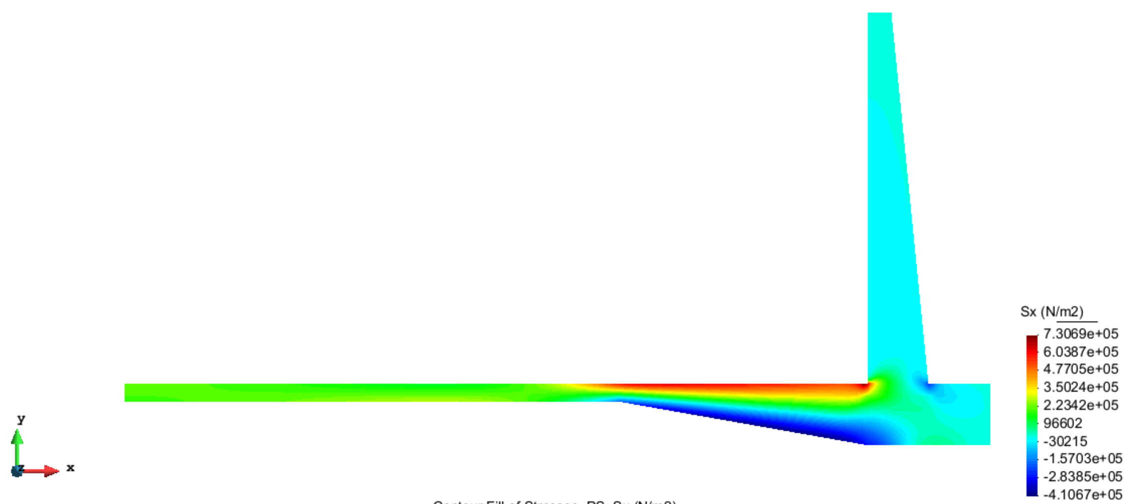
Contour Fill of Displacements, Disp-X (m).

The displacements in Y direction show that the base of the dam is the one more compressed.

Contour Fill of Displacements, Disp-Y (m).

Stresses:

For the stresses in the X direction show that the part of the structure which suffers more damage is the one close to the angle, which makes sense because is the zone where the two loads converge.



Contour Fill of Stresses_PS, Sx (N/m2).

The next figure shows the stress in Y direction in which we can see that the part that suffers the most is just the angle where the distributed load on the base and the linear distributed load on the wall converge.

Contour Fill of Stresses_PS, Sy (N/m2).