Master on Numerical Methods in Engineering

Computational Structural Mechanics and Dynamics

Practice 1

Elements definition and mesh refinement

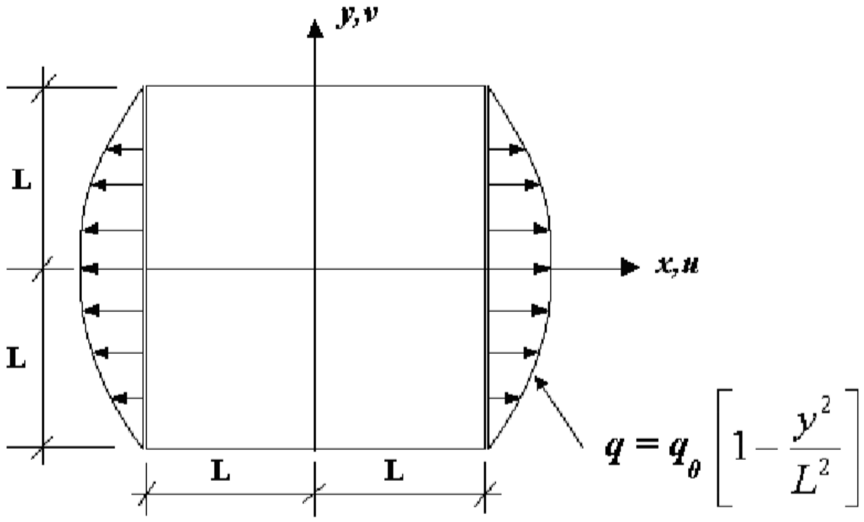
GiD and RamSeriess module

Mónica Ortega Castro

Exercise 1: Analysis of a thin plate under parabolic tensile force

Analyse the thin plate shown in the figure, which is submitted to a parabolic load. Compare the obtained results with the solution that is obtained when refining the mesh. Use triangular elements with 3 and 6 nodes and quadrilaterals with 4, 8 and 9 nodes.

Use symmetry conditions to simplify the problem.



Data

Material $\begin{cases} E = 2.1e5\ \text{MPa} \\ \nu = 0.30 \\ \text{Thickness} = 0.10\ \text{m} \end{cases}$

$L = 1.0\ \text{m}$

$q_0 = 100 \frac{\text{MN}}{\text{m}}$

The aim of the problem is to analyse the structure behaviour when varying (Table 1):

- Different discretization elements
- Shape function (number of element's nodes)
- Mesh refinement

Case	Element type	Number of nodes	Quadratic type
I	Triangular	Three nodes	Normal
II	Triangular	Six nodes	Quadratic
III	Quadrilateral	Four nodes	Normal or linear
IV	Quadrilateral	Eight nodes	Quadratic
V	Quadrilateral	Nine nodes	Quadratic9

Table 1. Cases number assignation to problem description

Problem solving:

As the problem description requires, symmetric conditions are applied as Figure 1 shows where point 1 is defined as a fixed point while line 1 is defined as 1dof line only in the y axis and line 2 in the x axis. Parabolic load is applied to line 3. The parabolic load was made as a product of two linear functions which product is equal to the **q** given equation.

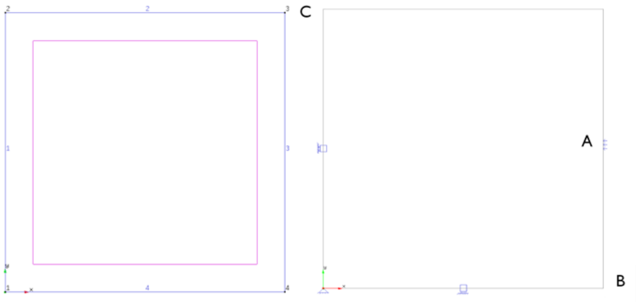


Figure 1. a) Points and lines labelling (pre) b) Boundary conditions description (post)

q load is defined is the only applied load and its defined normal outward 3 line, this structure submitted to a traction load is made out of steel. No self-weight is considered. Plane stress theory is applied.

It was decided to use an unstructured mesh definition. Columns "Mesh description" and "element size" are the common parameters in the simulations that define the column "Cases". This, with the aim of comparing amongst mesh refinement for the same element as well as from different cases.

Tables 3 (triangular elements) and 4 (quadrilateral elements) quantitatively compares x and y resulting displacement difference. This will define the most efficient case in terms of computational cost while assuring good results. Case study b (mesh description by default) was dismissed for Table 4.

Displacements are compared at point A, B and C (Figure 1).

Case	Mesh description	Element size	Elements	Nodes	x-disp	y-disp	x-disp	y-disp
1a	Nurb surface: 0.2	0,2	178	117	0,0024548	0,00024843	0,0037087	-0,0010347
1b	By default	0,1	208	125	0,0024578	0,00027208	0,0037214	-0,0010527
1c	Line 3: 0.05 Lines 1,2,4: 0.1	0,05	854	459	0,0024564	0,00028654	0,003754	-0,0010547
11a	Nurb surface: 0.2	0,2	178	411	0,0024554	0,00029067	0,0037716	-0,00105556
11b	By default	0,1	208	457	0,0024552	0,00029096	0,00377	-0,0010555
11c	Line 3: 0.05 Lines 1,2,4: 0.1	0,05	854	1771	0,0024552	0,00029083	0,0037719	-0, 0010555

Table 3. Cases definition for triangular elements. Displacements at point A, B, C.

Case	Mesh description	Element size	Elements	Nodes	x-disp	y-disp	x-disp	y-disp
IIIa	Nurb surface: 0.2	0,2	117	145	0,0024549	0,0002753	0,003754	- 0,00105532
IIIc	Line 3: 0.05 Lines 1,0.2,4: 0.1	0,05	260	292	0,0024556	0,00028796	0,0037664	-0, 001057
Iva	Nurb surface: 0.2	0,2	117	406	0,0024549	0,00029073	0,0037719	-0,0010554
IVc	Line 3: 0.05 Lines 1,0.2,4: 0.1	0,05	260	843	0,0024552	0,00029081	0,003772	-0, 0010554
Va	Nurb surface: 0.2	0,2	117	523	0,0024553	0,0002908	0,0037722	-0,0010555
Vc	Line 3: 0.05 Lines 1,0.2,4: 0.1	0,05	260	1103	0,0024552	0,00029082	0,0037722	-0, 0010554

Table 4. Cases definition for quadrilateral elements. Displacements at point A, B, C.

Figures 2 and 3 show the pre-processing from the project.

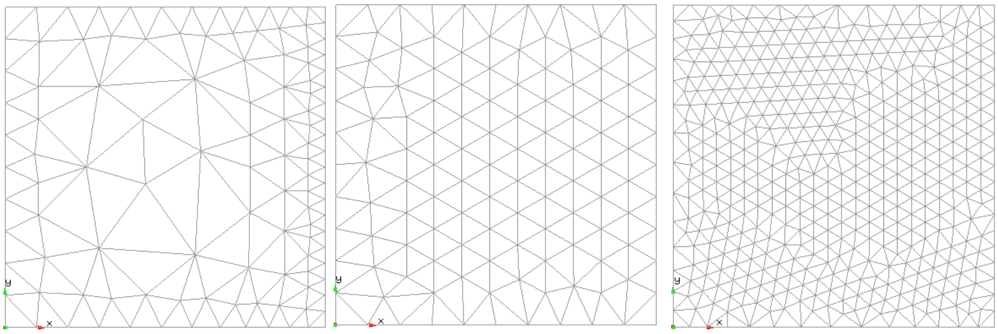


Figure 2. Mesh definition for cases *a, *b, *c (from left to right)

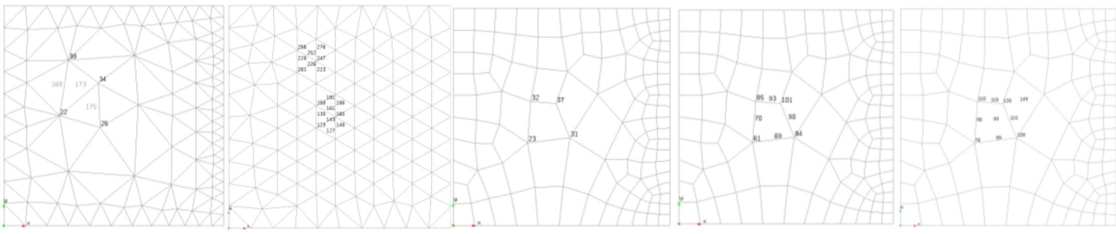


Figure 3. Shape function definition

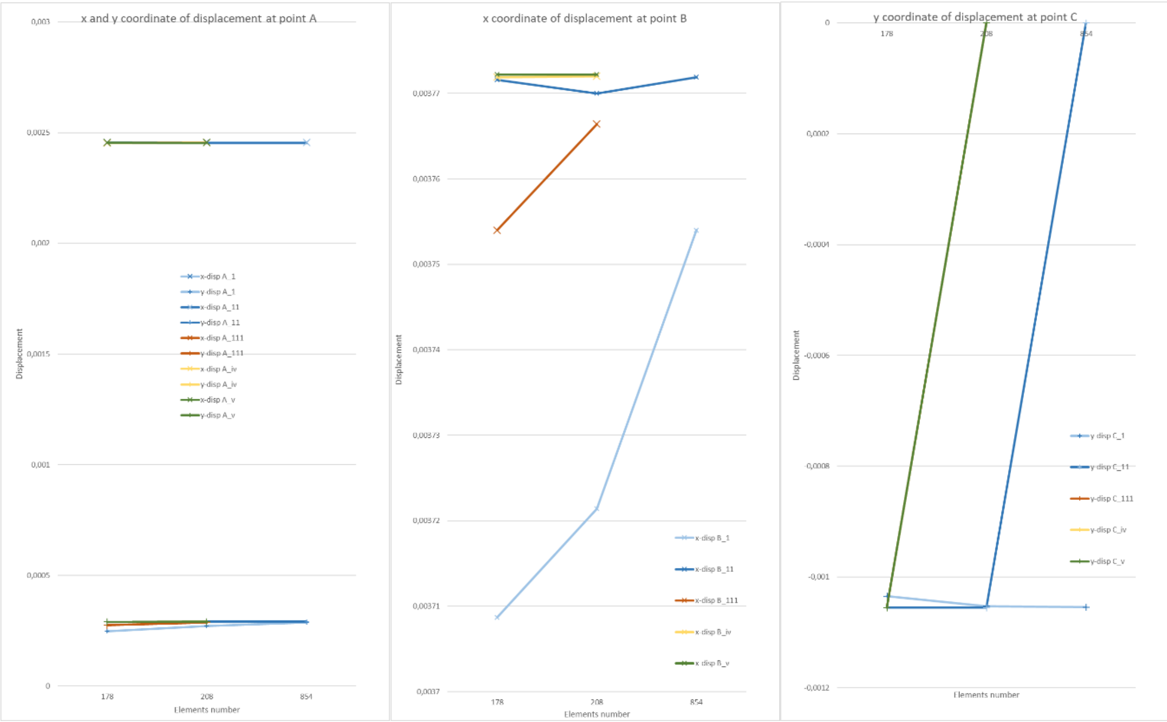
Conclusions

When solving this problem, it wwas decided to focus on the difference between the element types and mesh refinement. The parameter to compare with was the displacement at points A, B and C (Figure 1). The reasons to choose those points were because:

- Interest of learning the displacement of a point (A) where two degrees of freedom are expected.
- Interest of learning the displacement of a point (B) where one degree of freedom is expected, x-displacement.

- Interest of learning the displacement of a point (C) where one degree of freedom is expected, y-displacement.

Graphics from Figure 4 quantitatively represent the displacement value at those points.



Given a mesh description Table 5 and Table 6 compare displacement results for points A, B and C:

	a-type mesh description			
	x-disp A	**y-disp A**	**x-disp B**	**y-disp C**
	0,0024548	0,00024843	0,0037087	-0,0010347
	0,0024554	0,00029067	0,0037716	-0,00105556
	0,0024549	0,00027530	0,0037540	-0,00105532
	0,0024549	0,00029073	0,0037719	-0,0010554
	0,0024553	0,00029080	0,0037722	-0,0010555
Error difference	0,000001	0,00001	0,00001	0,0000001

Table 5. Minimum error difference between elements and shape functions

	c-type mesh description			
	x-disp A	**y-disp A**	**x-disp B**	**y-disp C**
	0,0024564	0,00028654	0,003754	-0,0010547
	0,0024552	0,00029083	0,0037719	-0,0010555
	0,0024556	0,00028796	0,0037664	-0,0010570
	0,0024552	0,00029081	0,0037720	-0,0010554
	0,0024552	0,00029082	0,0037722	-0,0010554
Error difference	0,0000001	0,00001	0,00001	0,000001

Table 6. Minimum error difference between elements and shape functions

Related to element types comparison: triangle has bigger errors than quadrilaters. It can be notice in both tables.

Related to same mesh description type: mesh description **c** is more precise for some displacements.

The decision pf which one to choose should depend on the aim of the project and accuracy required. There are not big differences for the current problem assignment.

The problem was pre and postprocessed in the FEM software GiDHOME while the used solver is RamSeries Educational 2D→Plane stress theory.

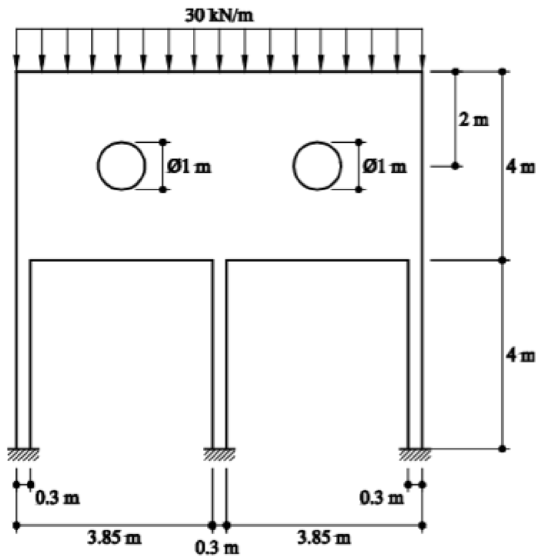
Exercise 11: Plate with two sections

The structure in the figure presents a reinforced concrete plate with two holes, supported by three columns. The central column undergoes a displacement δ due to sag of the foundation caused by a leakage in some pipes nearby.

Analyse the distribution of the stresses that the drop of the central column produces.

- **Case I**: Dead weight + Uniform load
- **Case II**: Dead weight + Uniform load + Settlement of the central column.

Assume the hypothesis of plane stress. Use triangular elements with 3 nodes for the analysis.



Data

$E = 3.0e10 \frac{N}{m^2}$

$\gamma = 25000 \frac{N}{m^3}$

$\nu = 0.2$

$t = 0.20\ m$ (Thickness of the plate and the columns)

Problem solving

Distribution of stresses is going to be analysed for both cases, under project conditions (no settlement) and due to the leakage effects in pipes nearby. To the latter, three cases were compared. Cases of study are defined in Table 1.

Caso	u_y (cm)	Description
Caso 1	0	Dead weight + Uniform load
Caso 2.1	1	Dead weight + Uniform load + Settlement of the central column
Caso 2.2	2	
Caso 2.3	10	

Table 1. Study cases to evaluate

Vertical displacement values (u_y) at middle column were realistic values to choose.

According to the geometry and load distribution, symmetric conditions w.r.t. axis Z could be applied. However, full 2D geometry was used to solve the problem (Figure 1).

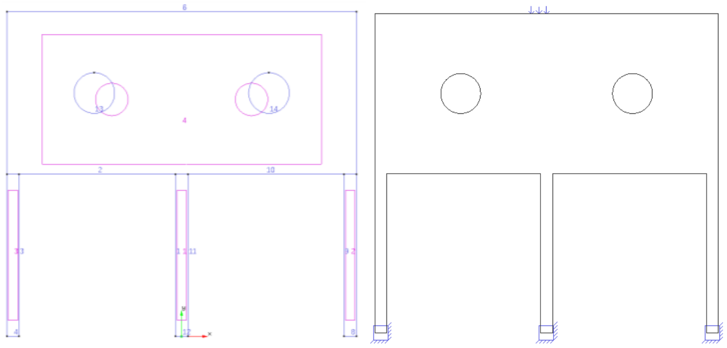


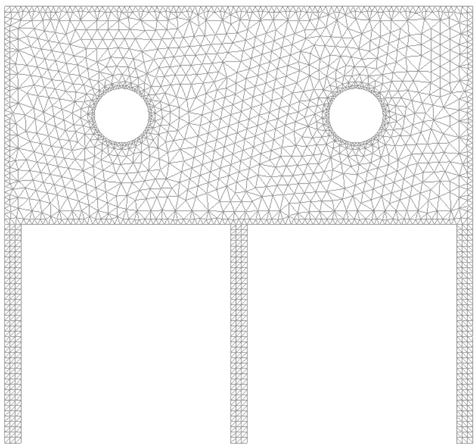
Figure 1. Geometrical description of the problem and conditions

Pre-processing highlights

Taking Figure 1 as the reference geometry the following conditions were settled:

- Case 1 lines 4, 8 and 12 were defined as fixed constraints for X and Y direction while for Case 2 line 12 is settled only in X direction and Y displacement varies from 1 to 5 cm according to Table 1.
- Line 6 is set as uniform static distributed load, 30KN/m, normal inward direction.
- Body weight is considered.
- NURB surfaces 1, 2 and 3 belong to pillar surfaces while surface 4 represents the remaining surface area, it includes the two inner holes. The surface selection was stated as the most appropriate configuration for meshing.

Mesh definition (Figure 2): According to problem definition, geometry is meshed by 3-noded triangle elements. After a mesh sensitivity analysis, it was decided to use an unstructured mesh with element size 0.2 m. Boundary lines (lines 3, 4, 6, 13 and 14) are refined as 0.1m.



Elements	3489
Nodes	2054
Element size	0,2
3-noded triangle elements	

Figure 2. Mesh definition

Discussion

- Displacements: Following four figures represent the displacement analysis for each case. Indeed, this part of the analysis is only used for the modeller because it verifies the expected displacement values. When comparing results, in not settlement considered, main displacement occurs at the upper structure and happens as a consequence of the distributed load distribution along line 6 towards the rest of the structure. Load is slightly and uniformly distributed in columns so that, displacements. However, there is a null displacement at the bottom of the pillars.
- This situation is complementary the opposite when settlements is assigned to nurb surface 1 (central pillar). Qualitatively, high displacement values is reorganised and mid-pillar absorbs most of the distributed load affection and convey it to the soil.
  Quantitative values are analysed in the next page.

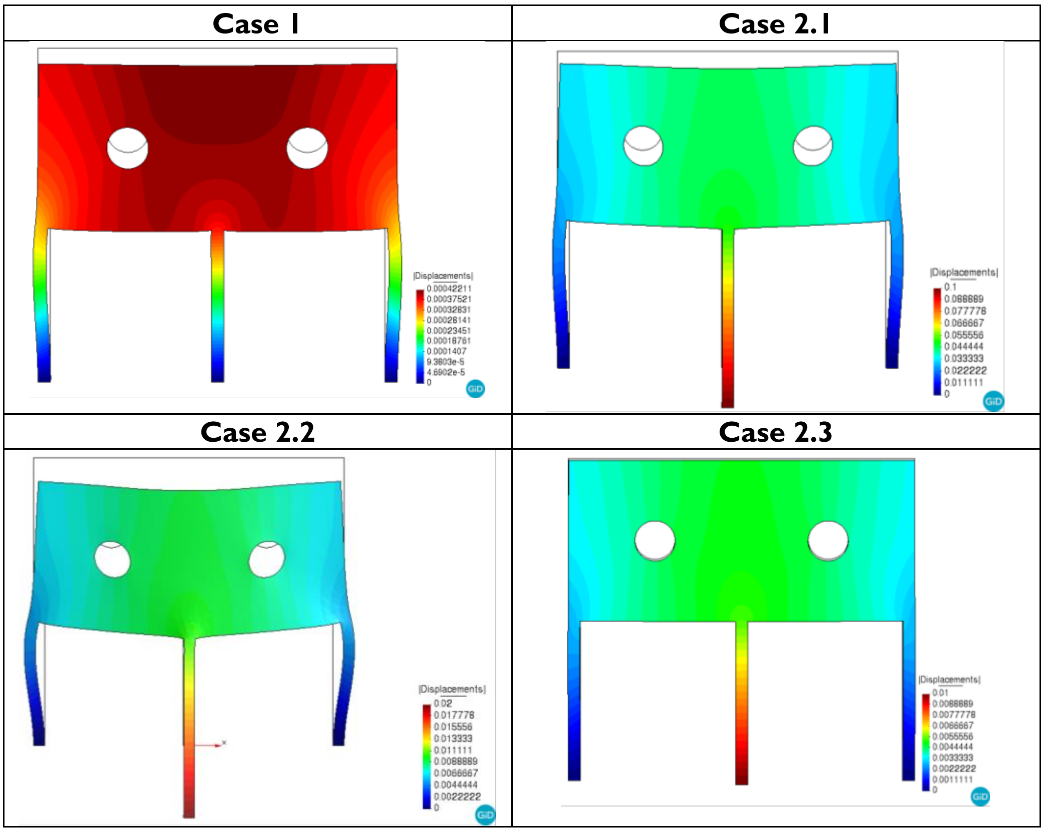


Figure 3. Displacement comparison. Not same scale factor applied

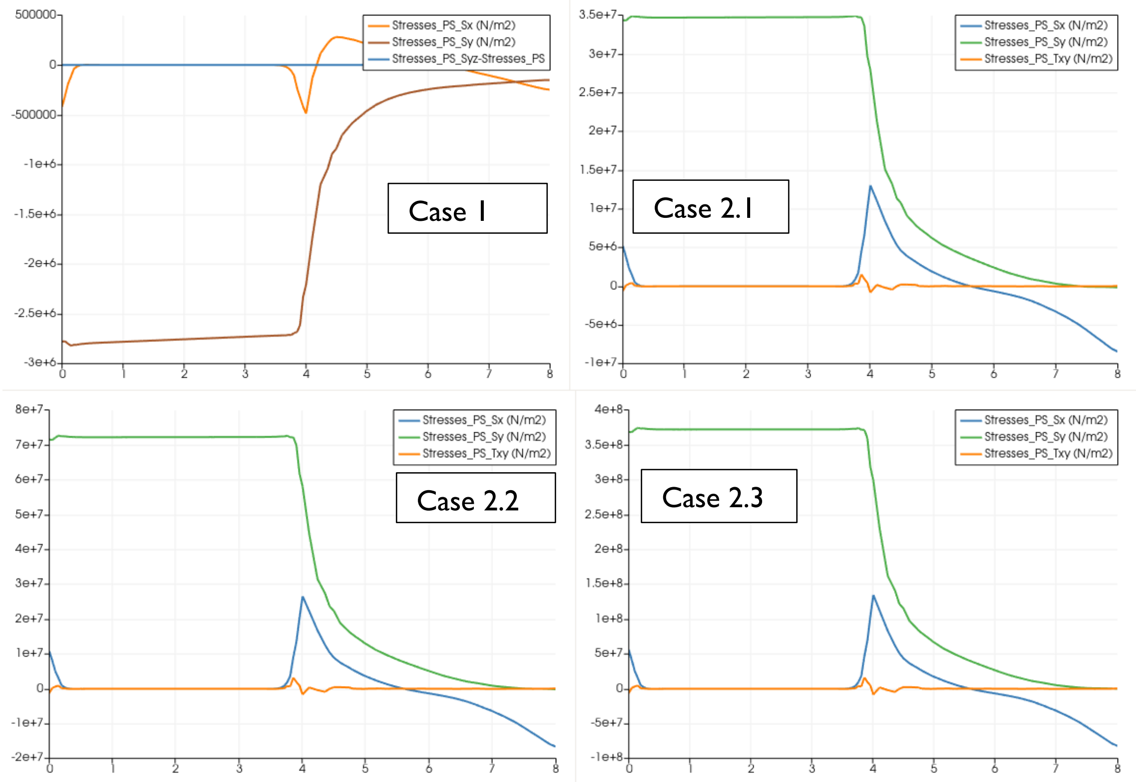


Figure 4. Stress components over line for all cases. Result values from points (0 0) and (0 8) were plotted

Figure 4 shows that stresses have same behaviour for cases 2.*. Only magnitudes are variable.

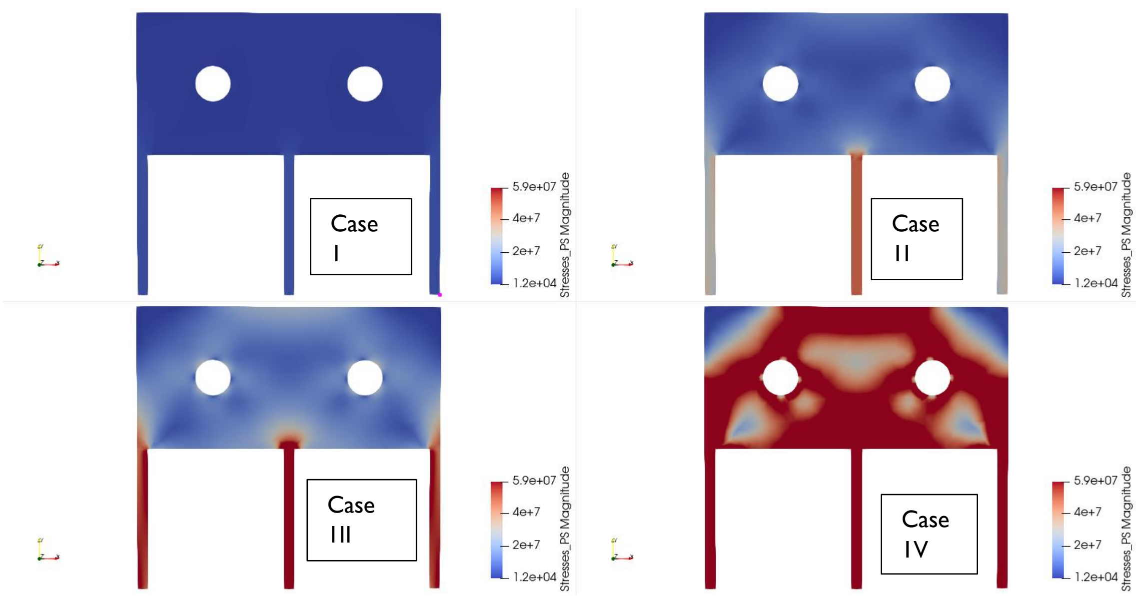


Figure 5. Stress magnitude for all cases.

As it was expected, displacement imposed in mid-column causes traction effects due to sag foundation while in case 1all columns are evenly subject to compression effects.

Figures 6, 7, 8 compare stress components. Consider that blue colour also includes negative values even when the shown scale is only positive. This happens to compare all simulations at the same scale.

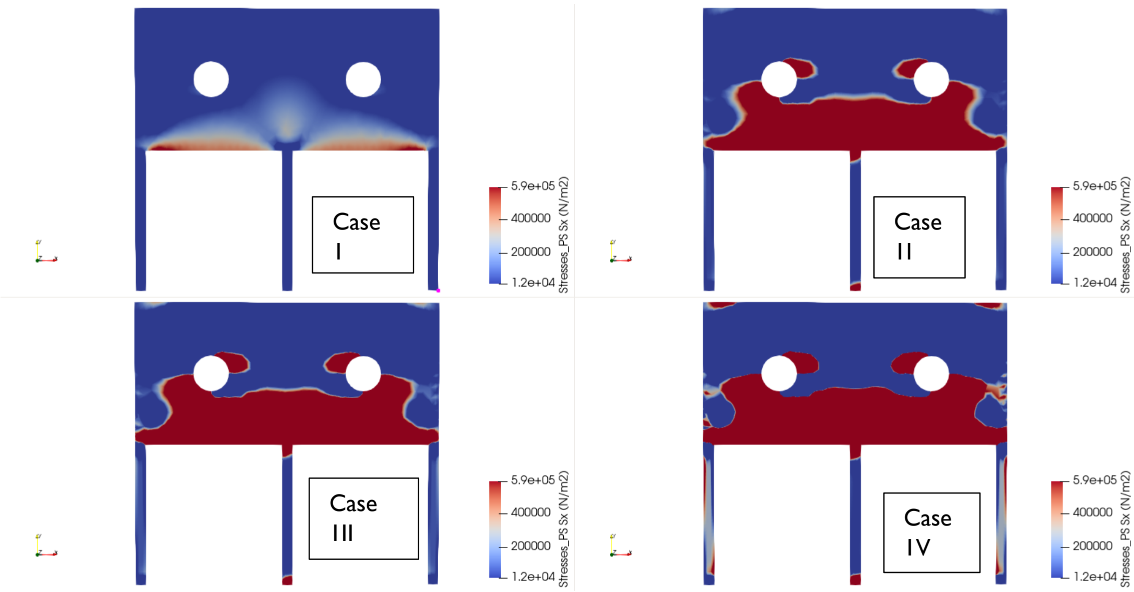


Figure 6. $\sigma_x$for all cases.

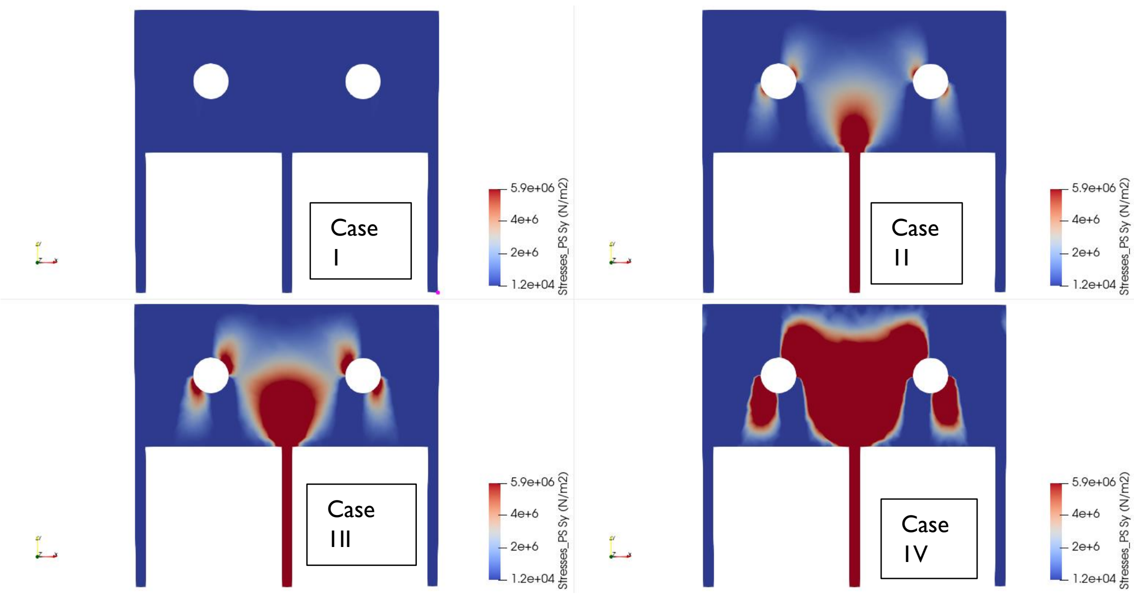


Figure 7. $\sigma_y$for all cases.

Lateral columns are forces to take/bear load from the mid-columns.

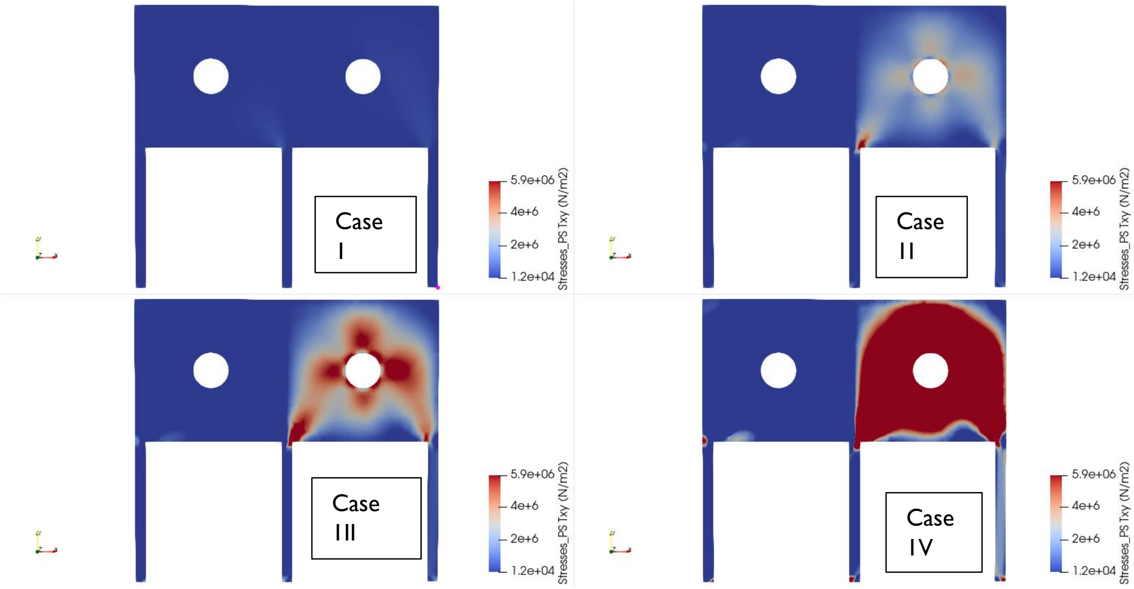


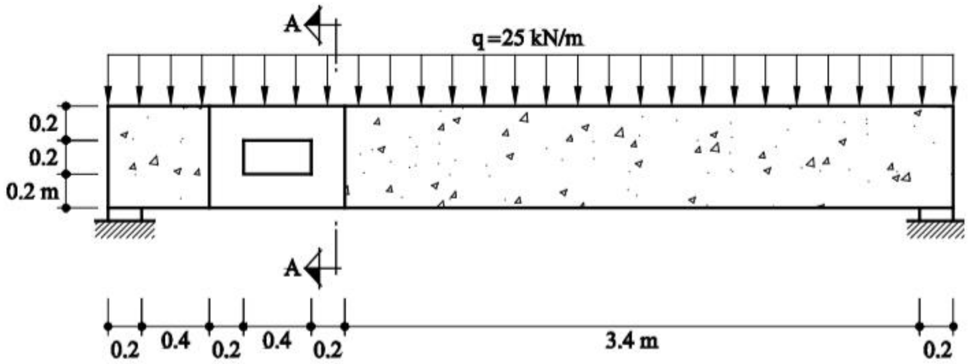
Figure 8. $\tau_{xy}$for all cases.

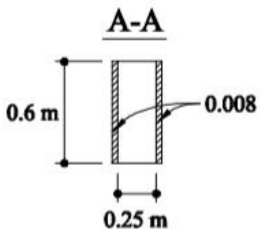
Even when numerically it seems to happen that there is no problem with the 10cm deformation. Tolerance studies should be carried out before taking any further decision.

The problem was pre and postprocessed in the FEM software GiDHOME while the used solver is RamSeries Educational 2D→Plane stress theory. Post-process was done with Paraview

Exercise 3: Plate with ventilation hole.

The structure represents a reinforced concrete plate with simple supports. This plate possesses a hole for a ventilation pipe. Due to a change in the initial project, the design load for which the plate was calculated increased significantly. This motivated the placement of a metal reinforcement sheet on both sides of the plate in the area of the hole. Analyse the state of stress in the plate and the metal reinforcement sheets. Assume the plane stress hypothesis. Use structured mesh of quadrilateral elements with four nodes.





Data

$$\text{Concrete}\begin{cases} E = 3.0e10\,\dfrac{N}{m^2} \\ \nu = 0.2 \\ t = 0.25\text{ m} \\ \gamma = 25000\,\dfrac{N}{m^3} \end{cases} \qquad \text{Steel}\begin{cases} E = 2.1e11\,\dfrac{N}{m^2} \\ \nu = 0.3 \\ t = 0.016\text{ m (Two sheets of 0.008m)} \\ \gamma = 78000\,\dfrac{N}{m^3} \end{cases}$$

Pre-processing:

Following conditions and mesh refinement (Figure 1) were set according to the problem definition.

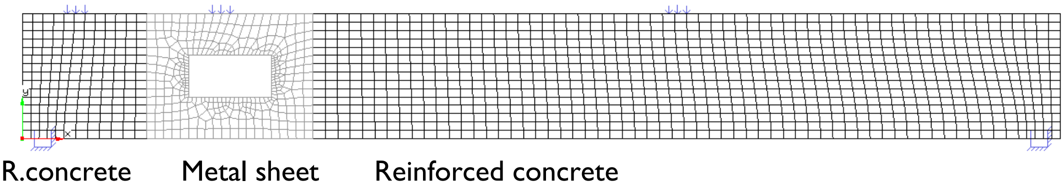


Figure 1. Mesh description

Elements	Nodes	Element size	Comment
1459	1635	0.06	Mesh refinement considered

Table 1. Mesh definition to use

Self-weight is not considered.

Results analysis

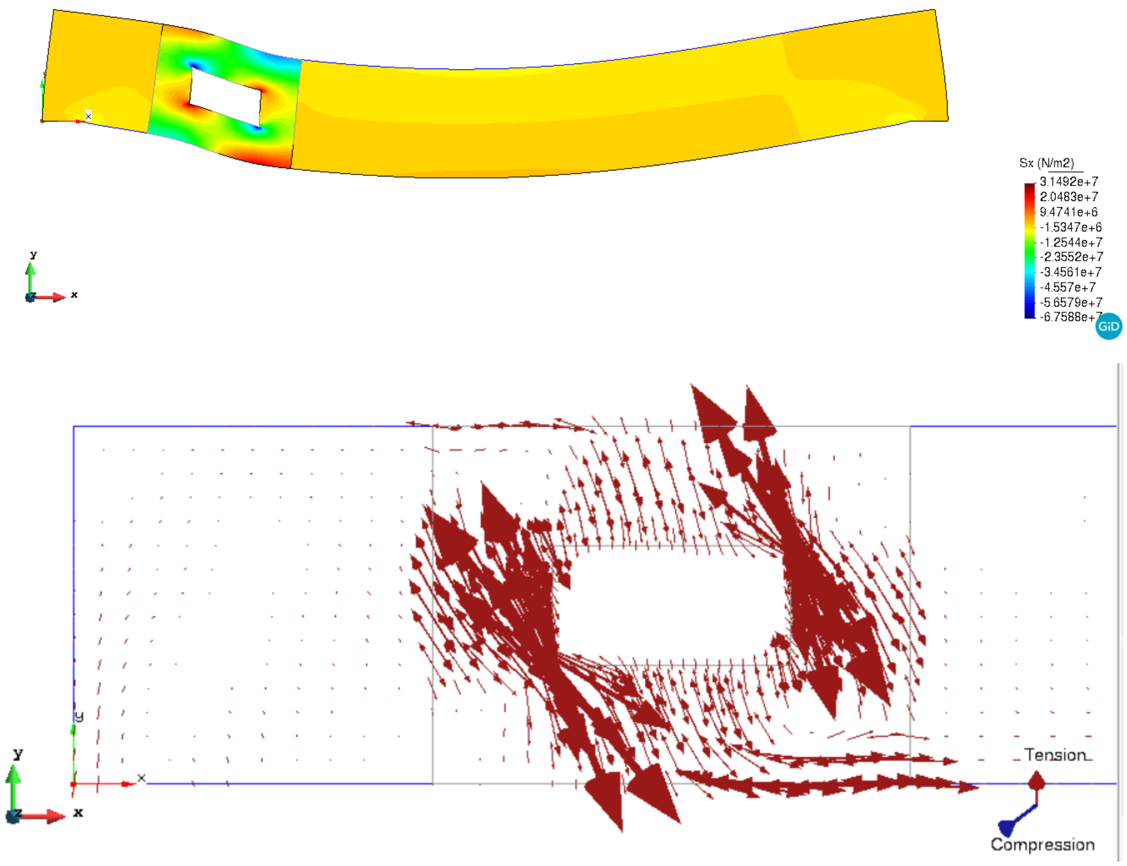


Figure 2. σ_x. Result surface (upper image) and display vectors (lower image)

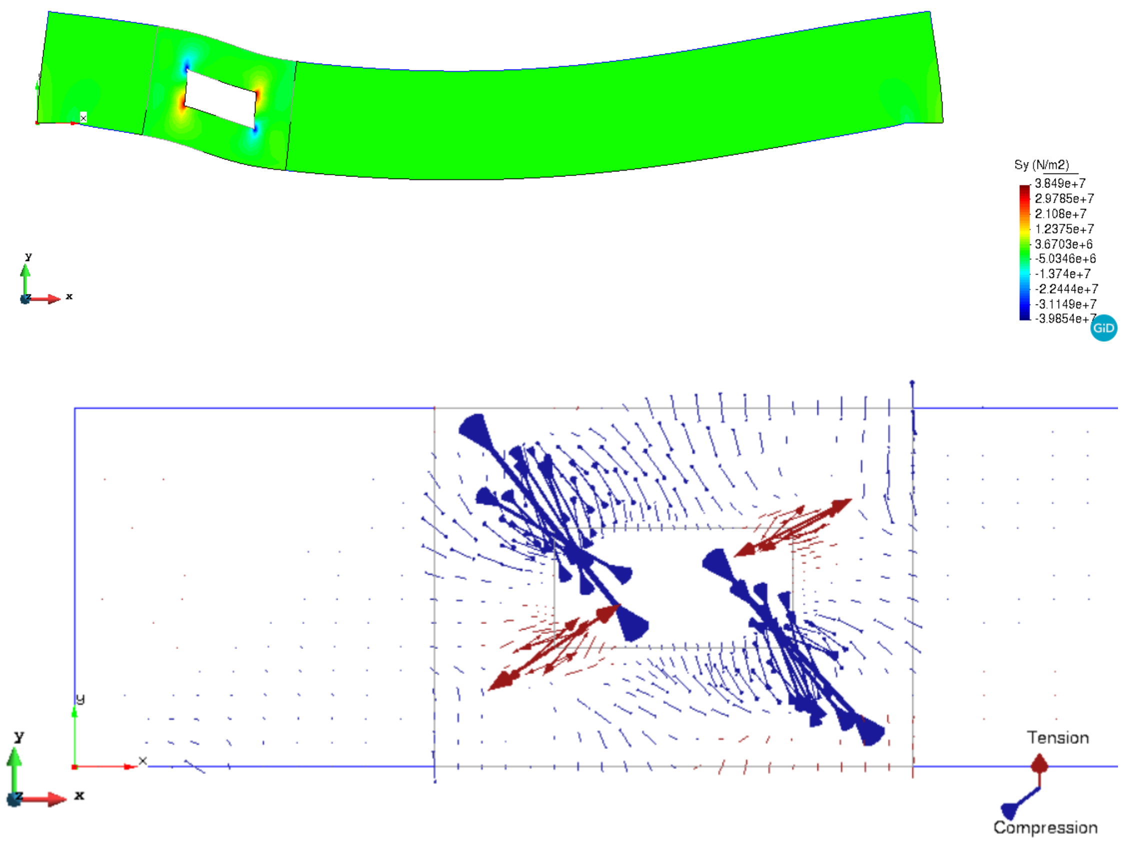


Figure 3. σ_y Result surface (upper image) and display vectors (lower image)

<u>Conclusions</u>

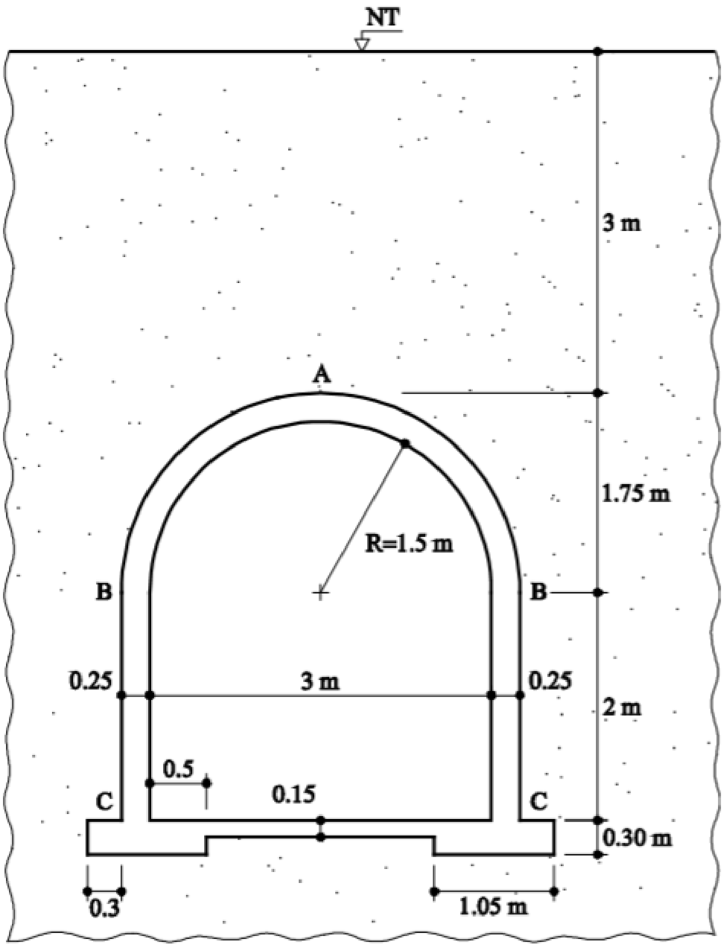
Reinforced concrete is submitted to compression loads. However, metal is suffering from both, but above all traction as it is expected.

Deeper study should be conducted to take a decision.

Exercise 4: Tunnel

The structure shown in the figure represents the cross-section of a tunnel made of reinforced concrete. The tunnel is used by the oil industry to transport sunflowers from a warehousing silo to the processing plant.

Analyse the state of stress in the cross-section of the tunnel, considering that the base slab is elastically supported by the ground. Use the hypothesis of planar deformation. Use quadrilateral elements with four nodes.



Data

Concrete $\begin{cases} E = 3.0e10 \dfrac{N}{m^2} \\ \nu = 0.2 \\ \gamma = 25000 \dfrac{N}{m^3} \end{cases}$

Ground pressure $\begin{cases} P_A = 5.4e4 \dfrac{N}{m^2} \\ P_{BC} = 2.16e4 \dfrac{N}{m^2} \\ P_{AB} = \text{linear variation} \end{cases}$

Load coefficient of the ground $= 50 \dfrac{N}{cm^3}$

According to the problem definition, plane strain theory will be applied. Self-weight is no considered.

Figure 1 represents the geometry and mesh definition. Loads applied are represent by arrows.

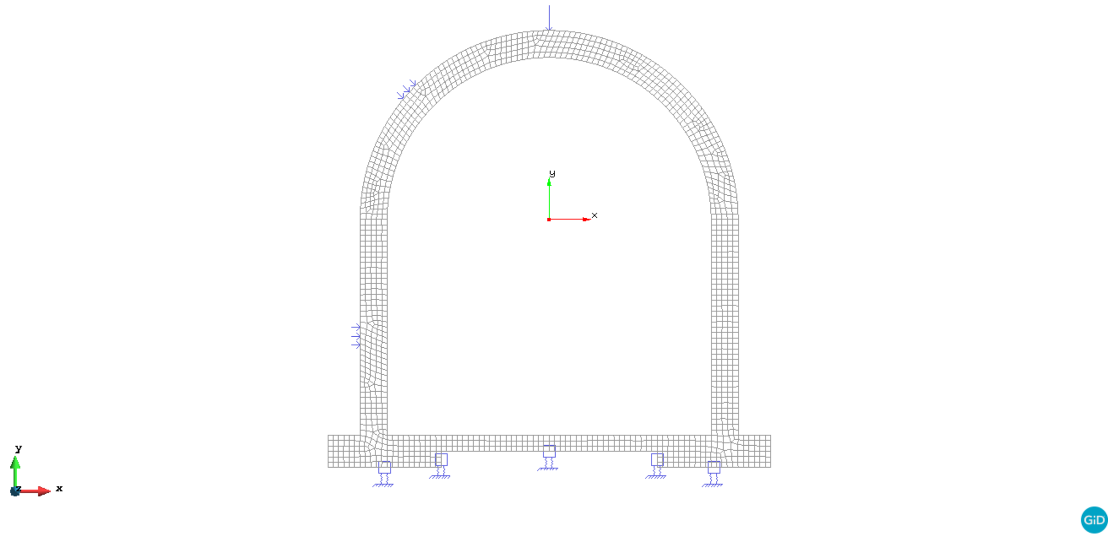


Figure 1. Mesh definition

Elements	Nodes	Element size	Comment
1298	1568	0.29	Unstructured mesh considered

Results:

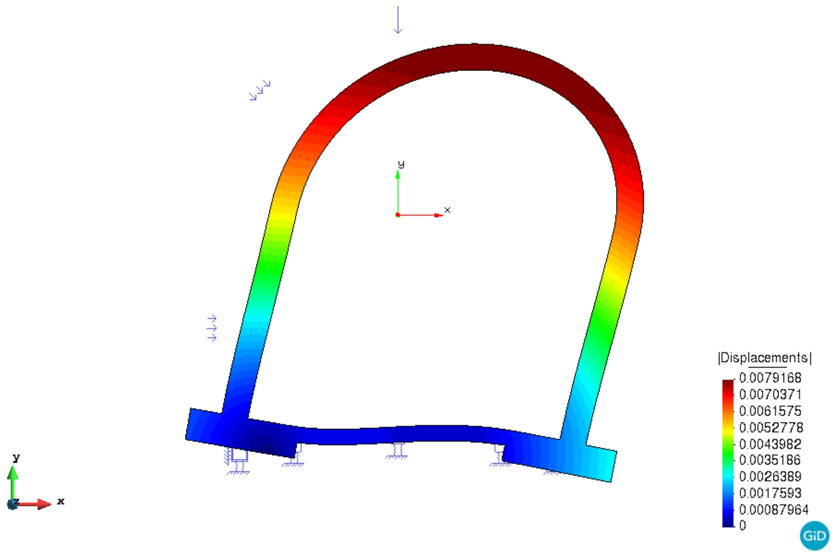


Figure 1. Displacement module. Result surface filter

It is observed that loads exert higher displacement on the tunnel vaulting than in the rest of the tunnel. Actually, displacement is lower than a millimetre, however, main displacement direction goes from left to right (Figure 1).

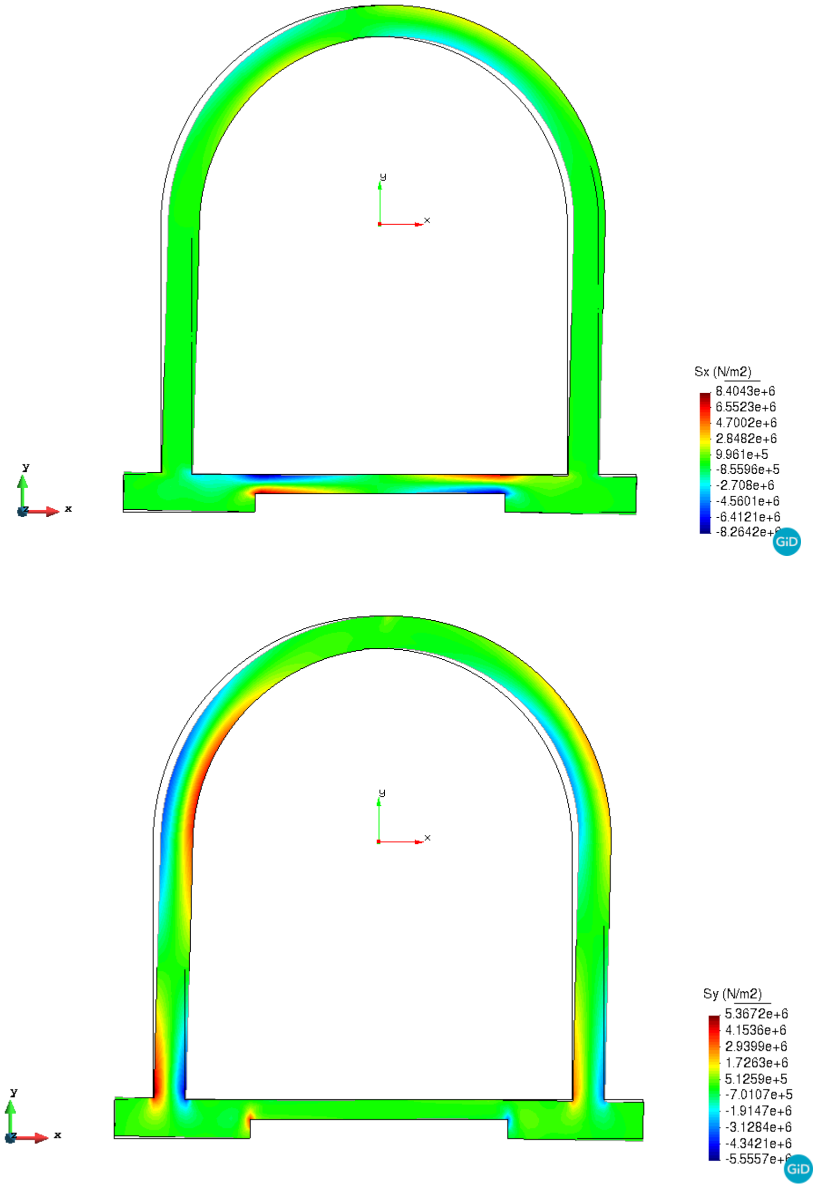


Figure 2. Stresses in x and y direction

The problem was pre and postprocessed in the FEM software GiDHOME while the used solver is RamSeries Educational 2D →Plane strain theory.