Master on Numerical Methods in Engineering

Computational Structural Mechanics and Dynamics

Practice 3

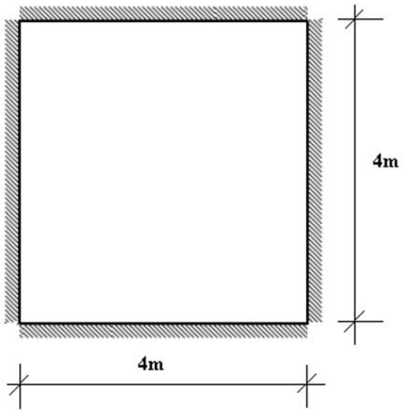
Plates

GiD and RamSeries module

Mónica Ortega Castro

Exercise 1: Clamped plate with a uniform load

Analyse the state of stress of the quadratic plate in the figure, whose four sides are clamped. The plate is submitted to a uniform load q. Use triangular plate elements DKT, triangular Reissner-Midlin elements with 6 nodes with reduced integration and quadrilateral elements CLLL for the analysis. Compare the obtained results for the deflection in the center of the plate with the analytical solution.



Data

Concrete $\begin{cases} E = 3.0e10\ \dfrac{N}{m^2} \\ \nu = 0.2 \\ t = 0.10m\ (\text{thickness}) \end{cases}$

Load $\left\{ q = 1.0e4 \dfrac{N}{m^2} \right.$

The aim of the problem is to analyse the numerical solution for the given elements in comparison with the analytical solution when comparing vertical displacement (the only one possible for this problem type) at the center of the plate (0,0).

Numerical solution:

Definition of conditions (Figure 1):

- Lines 1,2,3,4 suffer from a constrain for the 3 dof (vertical displacement and the two rotations).
- Material properties from concrete
- Uniform static load of value 1000 N/m2 inwards and normal direction to the plate

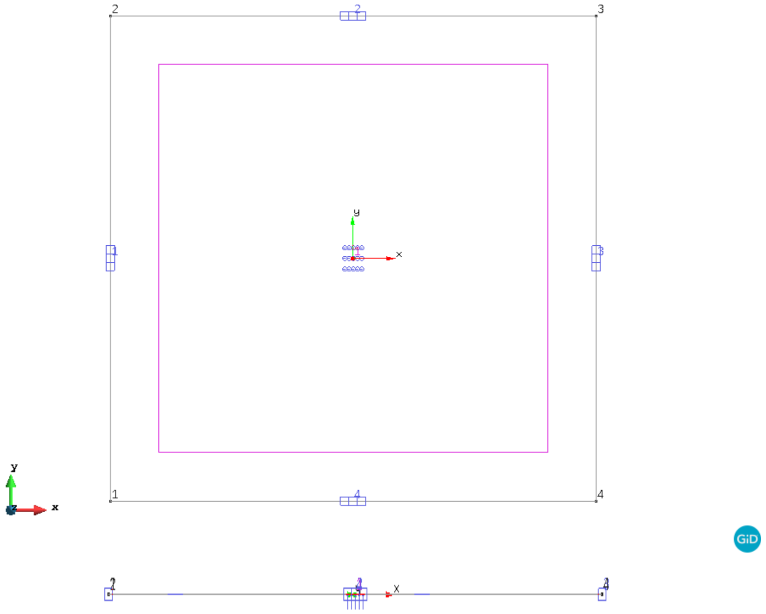


Figure 1. Problem conditions

Structured mesh definition:

Table 1 contains the plate discretization into the three element types and the meshes description.

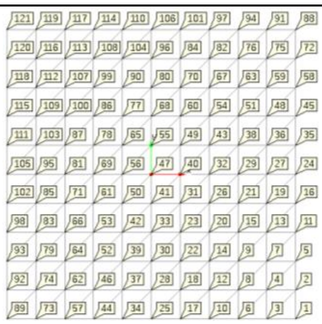
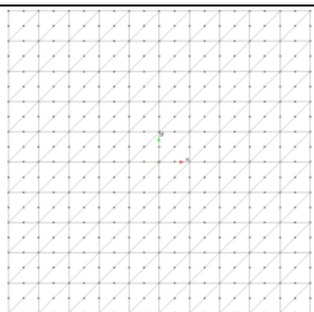
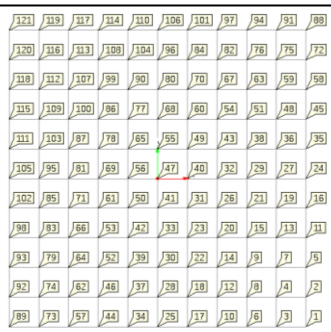
 Triangular plate elements (DKT)	 Triangular RM elements with 6 nodes with reduced integration (RM)	 Quadrilateral elements CLLL
200 elements	200 elements	100 elements
121 nodes	441 nodes	121 nodes

Table 1. Mesh description for the three given element types (element size 0.4)

FEM results:

Figure 2 and 3 represents the vertical displacement when numerically solving with each element type.

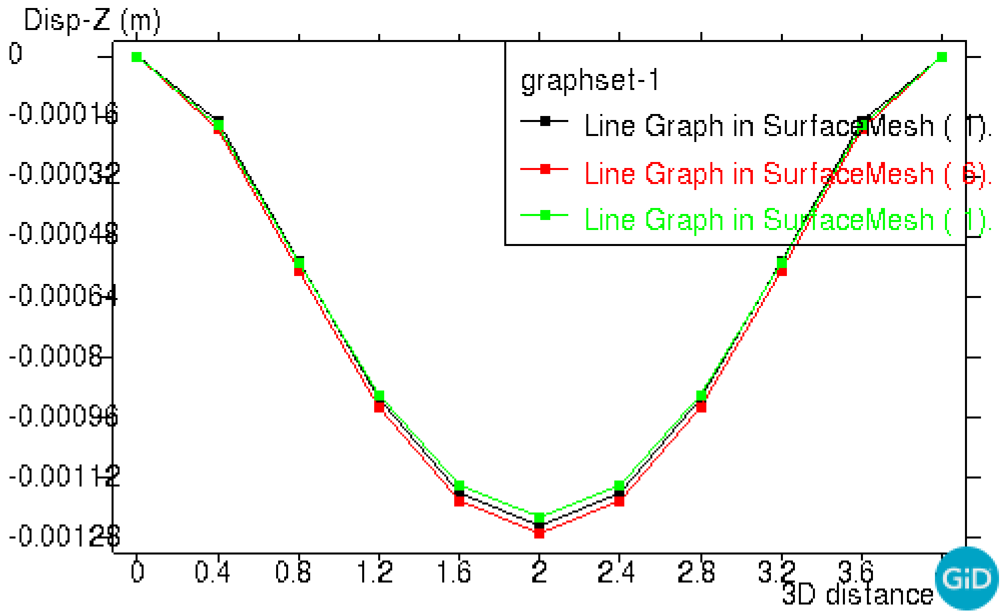


Figure 2. Vertical displacement along line in the middle of the place and crossing the (0,0) coordinate. (Colour legend: DKT element, CLLL element, DKT element).

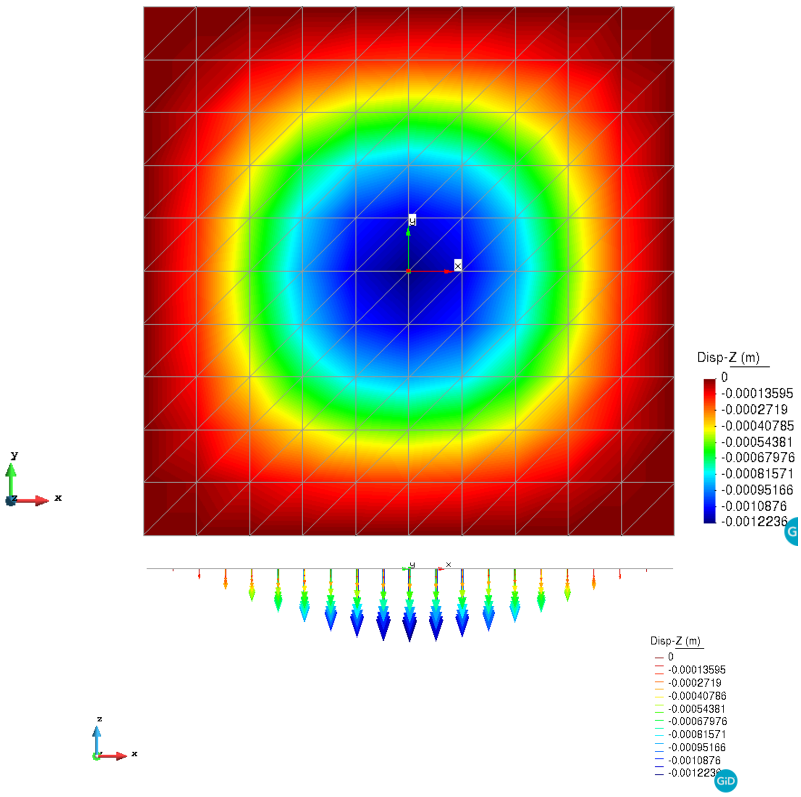


Figure 3. Vertical displacement results for RM element type. Upper image: surface results XY-plane. Lower image: display vector XZ-plane

Analytical solution

Maximum vertical displacement for a clamped plat submitted to a uniform load q that is normal to the surface is:

$$w_{max} = \alpha * \frac{q * L^4}{D}$$
$$D = \frac{E * t^3}{12(1 - v^2)}$$

where:

L: 4m (plate side length)
q: $1e4$N/m² (uniform load)
E: $3e10$N/m² (Young modulus)
v: 0.2 (Poisson ratio)
t: 0.1m (thickness)
$\alpha = 0.00126\ for\ square\ plates$[1]

This yields to

$$D = 2.604e6$$

$$w_{max} = 0.0012386m$$
Maximum displacement is at the center point (0,0)

Conclusions:

The idea is to compare vertical displacement between analytical and numerical solutions. This, with the aim to decide what is the most appropriate element type. All the other parameters are the same in order to let provide an appropriate comparison.

Solution	Element type	Max vertical displacement (Wmax)	Relative error
Analytical		0.0012386	0.00%
Numerical	DKT	0.0012672	2.31%
	RM	0.0012236	1.21%
	CLLL	0.0012474	0.71%

Table 2. Analytical vertical displacement in comparison numerical solution

For this case, a convergence tolerance of 1e-5 m was considered enough for the aim of the study. As bibliography sources indicates, quadrilateral elements (CLLL) provide more accurate results than RM elements and this one is more accurate than DKT element. Thus, following graphical results are shown only for CLLL element.

[1] Roark's Formulas for Stress and Strain Book 7th Edition. Values edited for concrete material (v =0.2)

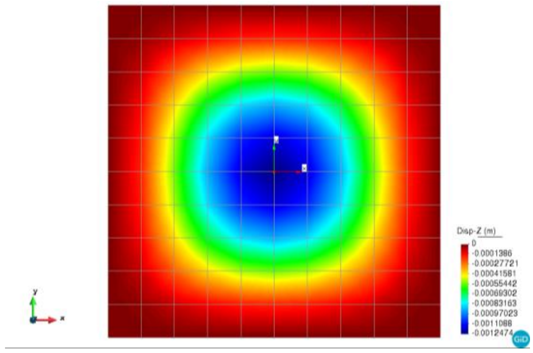


Figure 4. Vertical displacement (Result surface)

Compression is higher in the central area of the plate and maximum displacement is observer at the central point. It follows a symmetrical behaviour for geometry and loads distribution.

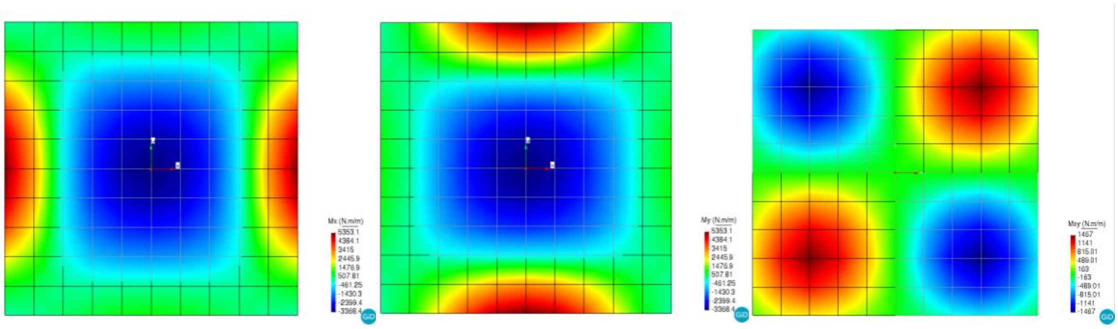


Figure 5. Bending momentum results (Result surface filter)

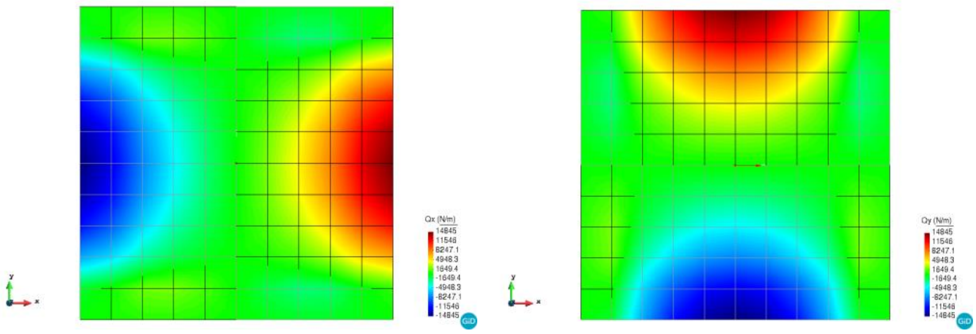
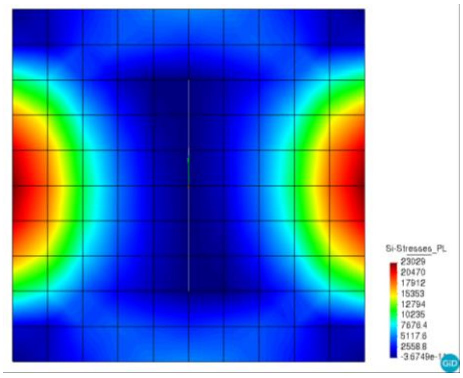
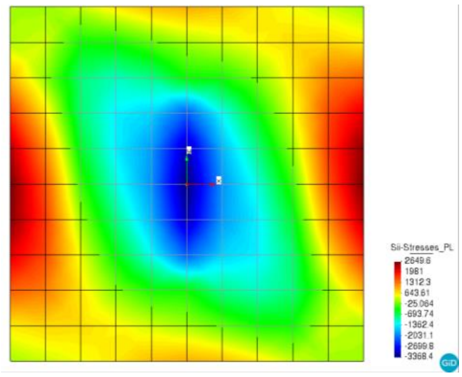


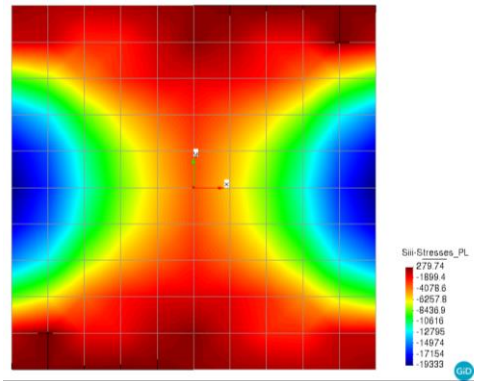
Figure 6. Shear forces results (Result surface filter)

Shear stress is observed at the extreme sides of the plate, in this case in the y-axis maximum while the minimum is at y-axis minimum value. Same behaviour addapted to x-axis is observed.

Symmetrical results are noticeable for stresses (Mx-My, Qx-Qy).





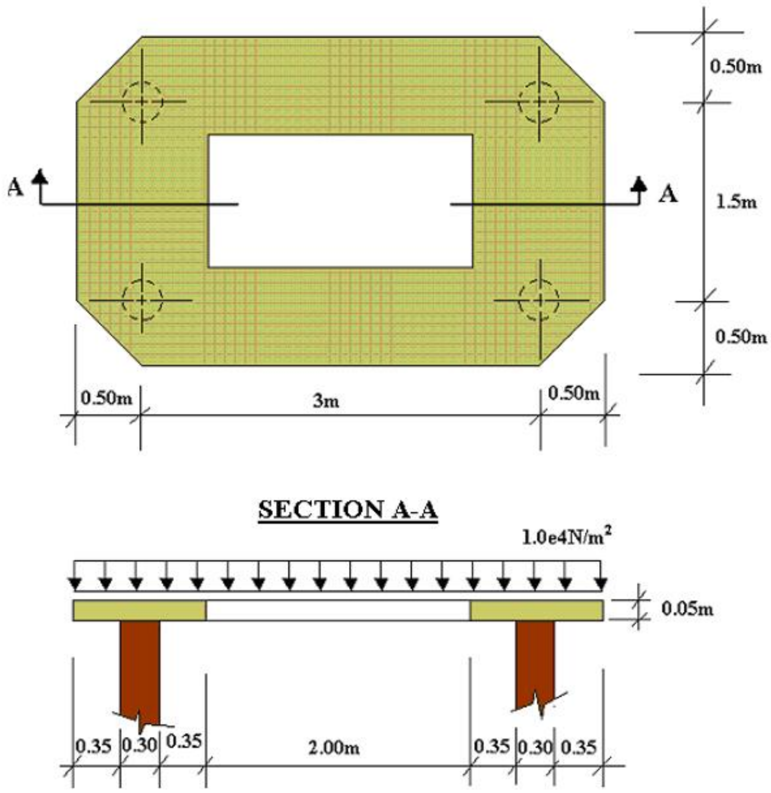


For the given clamped case, maximum shear occurs at the edges. When operating the structure, it should be guaraantedd that applied load doesn't cause superior stresses that are admissible for the material.

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

Exercise 1I: Exercise 2: Thin plate with internal hole

The figure shows a steel plate supported on four columns. Analyse the structural behaviour of the plate using the theory of thin plates. Use triangular elements DKT.



Data

$$\text{Steel}\begin{cases} E = 2.1e11\,\dfrac{N}{m^2} \\ \nu = 0.3 \\ \gamma = 7.80e4\,\dfrac{N}{m^3} \end{cases}$$

In this exercise it is worth mentioning the pre-process settings. Three options/assumption types were considered:

1. 3 dof contrainst on the pillars nurb surfaces. Material and static load properties applied to both surface types, pillars and plates. Surface discretization for pillars and other for steel plate.
2. 1 dof constrain related to vertical displacement allowing only rotations. This is because vertical displacement due to compression is expected to be low. Material and static load properties applied to both surface types, pillars and plates. Surface discretization for pillars and other for steel plate.

3. Steel material properties for plate and concrete for the pillars (1m normal direction). Steel surface is fully defined, and pillars surface are underneath. Hard to reproduce this option. Pillars vertical displacement constrained.

Second option was chosen thus, results will be represented for this case.

Geometry definition:

In total, 5 surfaces were created. 4 related to pillars and one related to the plate. Pillar surfaces were extracted from the latter as well as the mid-hole.

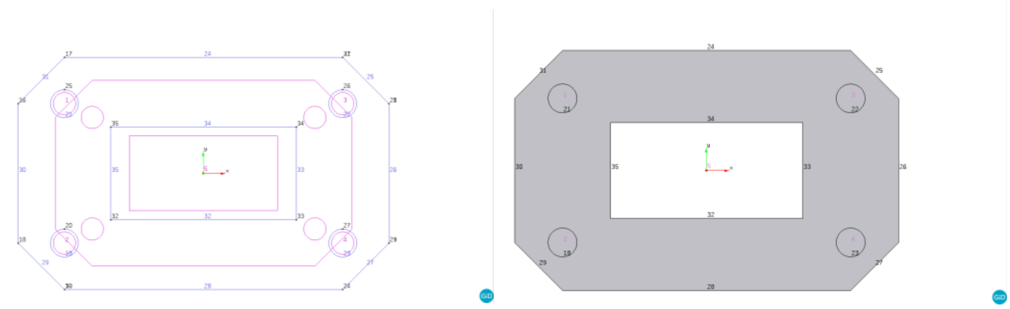


Figure 1. Geometry definition. L.hs. image: normal render. R.hs. image: flat render

Mesh definition:

It was chosen an unstructured mesh with DKT elements (3-noded triangular elements). Mesh refinement of 0.05 m were defined at the pillar surfaces.

Convergence evaluation (Figure 2) was executed in order to choose an accurate vertical displacement solution. Convergence tolerance was chosen as 1e-6 m.

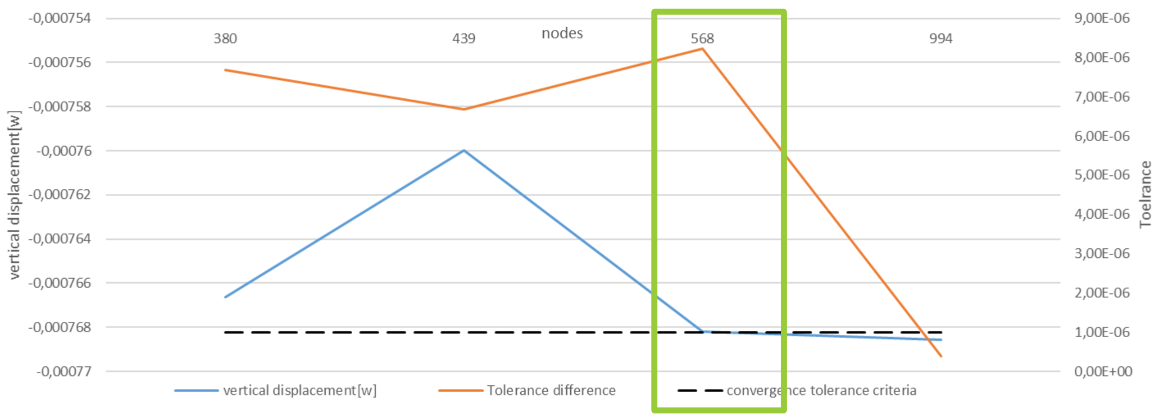


Figure 2. Convergence tolerance test

Chosen mesh description (Figure 3) is:

- Element size 0.15 m
- 1008 elements
- 568 nodes
- Vertical displacement -0.00076819 m

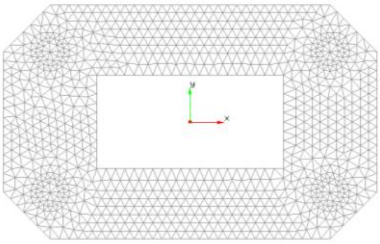


Figure 3. Mesh description

FEM results:

In order to run the simulation, these were the followed conditions:

- q load is applied is applied in gravity direction
- self-weight effects neglected.

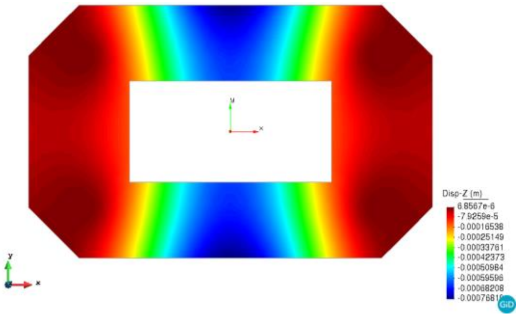


Figure 4. Vertical displacement. Surface results filter

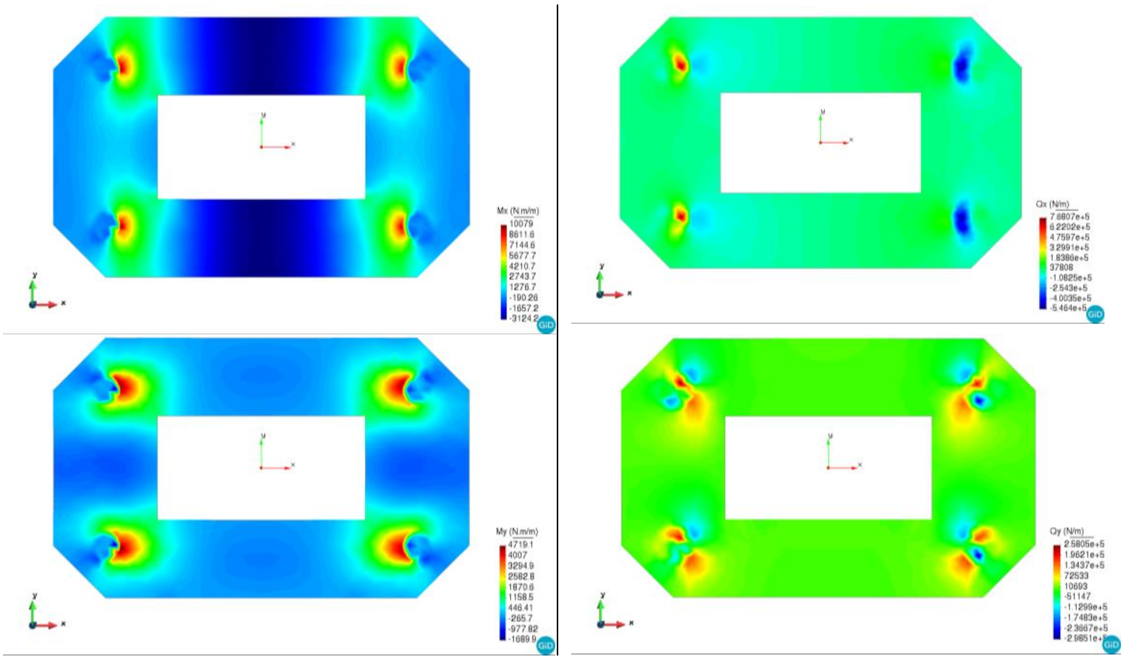


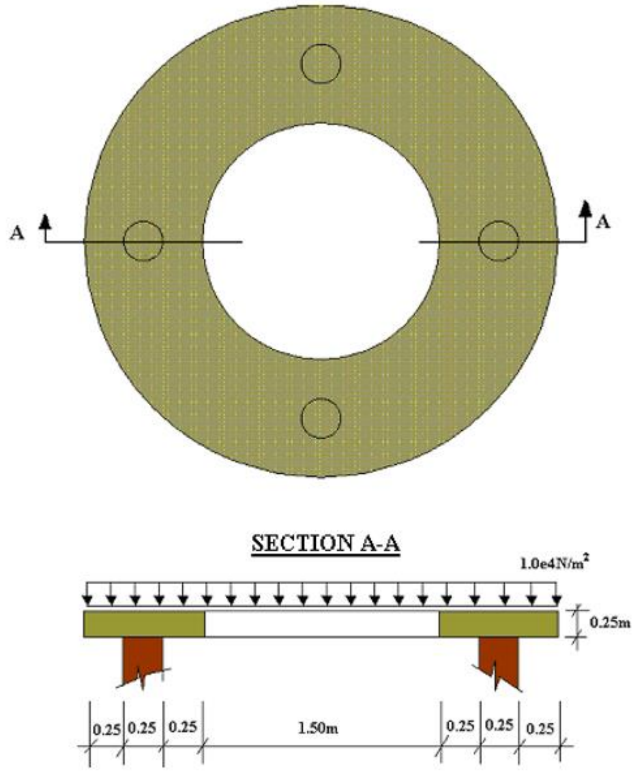
Figure 5. Bending moments (left hand side pictures) and stresses applied due to uniform load (left hand side pictures)

It is observed that in Figure 5 bending moments and stresses are symmetrical, first ones in x and y axis direction while the second one in x or y direction.

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

Exercise 3: Thick circular plate with internal hole

The figure shows a reinforced concrete plate supported on four columns, submitted to its dead weight and a uniform load. Analyse the structural behaviour using the theory of thick plates of Reissner-Mindlin. Use triangular elements of Reissner-Mindlin with six nodes and reduced integration.



Data

$$\text{Concrete}\begin{cases} E = 3.0e10\,\dfrac{N}{m^2} \\ \nu = 0.2 \\ \gamma = 2.4e4\,\dfrac{N}{m^3} \end{cases}$$

The aim of this project is to analyse structural behaviour of the structure by using the theory of thick plates of Reissner-Mndlin.

Geometry (Figure 1) and mesh refinement (Figure 2) description were defined by following same procedure as in Exercise 2. Simulation conditions can be observed in Figure 3. Convergence tolerance criteria of at least 1e-5m was chosen.

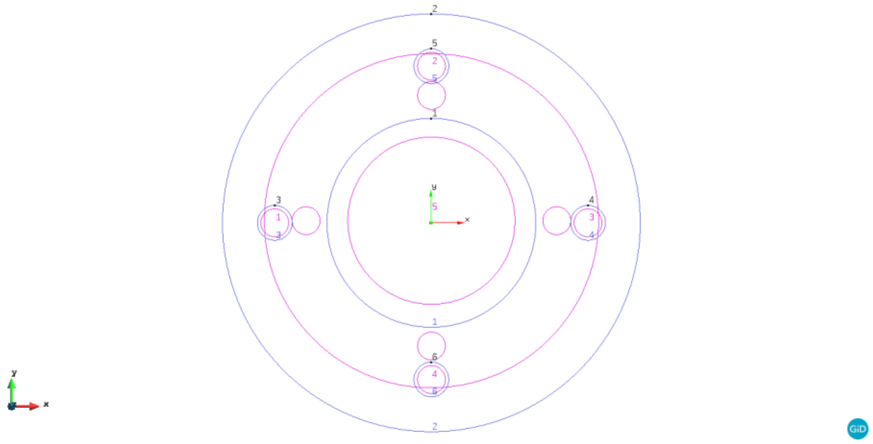


Figure 1. Geometry description

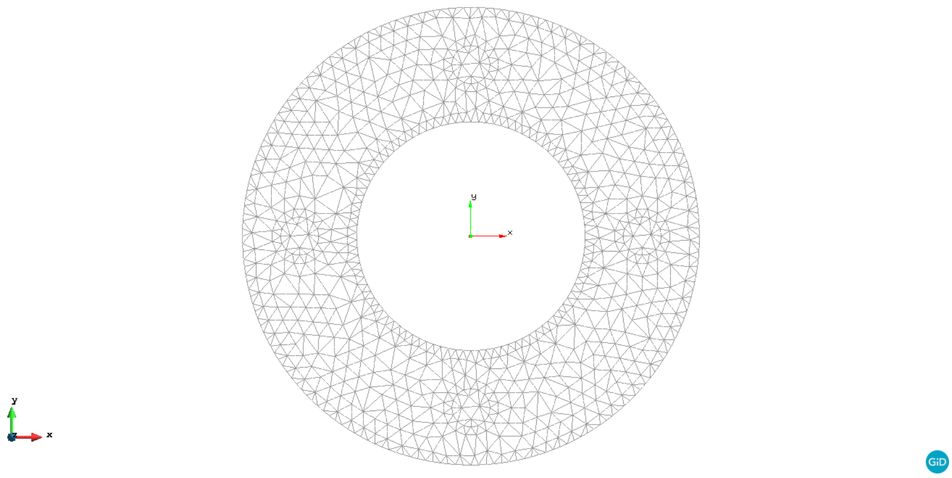


Figure 2. Mesh definition

Resulting mesh features are as follow:

- RM element
- 1574 elements
- 3430 nodes

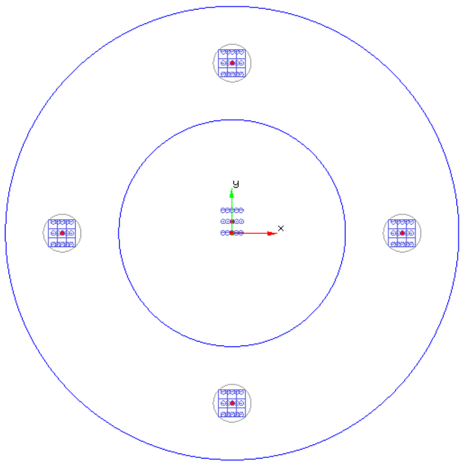


Figure 3. Fixed constraints applied

FEM results

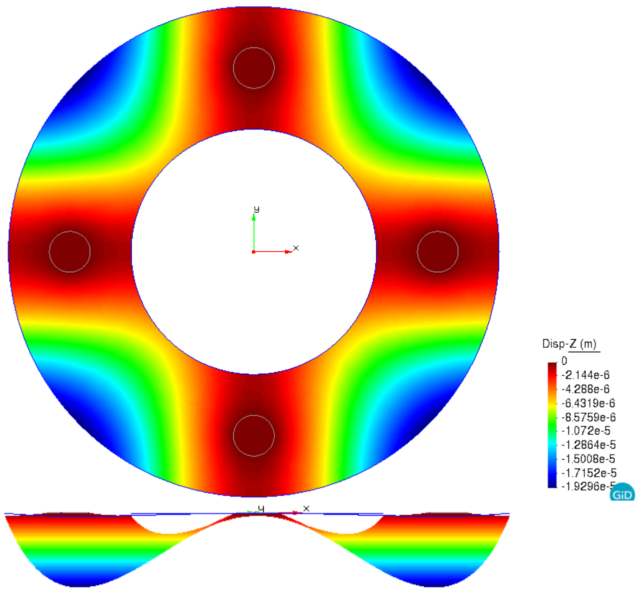


Figure 4. Vertical displacement and deformed structure. Result surface filter

Behaviour is symmetric. Some computational cost could have been saved by using symmetric conditions. High displacement values are located at the edged between two pillars and lowest values occur over the pillar surfaces. Deformation factor is 1000 which means displacements are small. This is because this is a thick plate case.

Figures 5 and 6 indicates a symmetric behaviour in terms of stresses and thus, in terms of bending moments too.

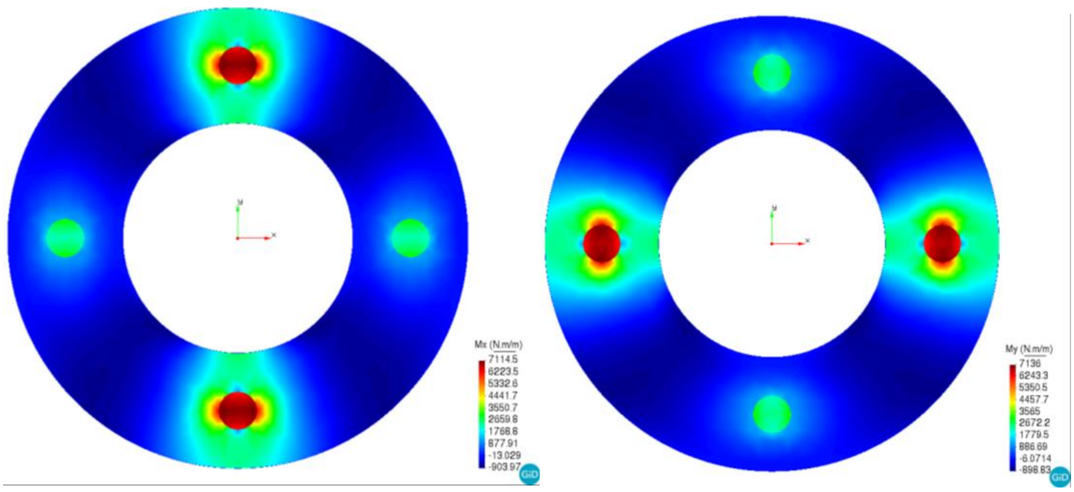


Figure 5. Bending moments result

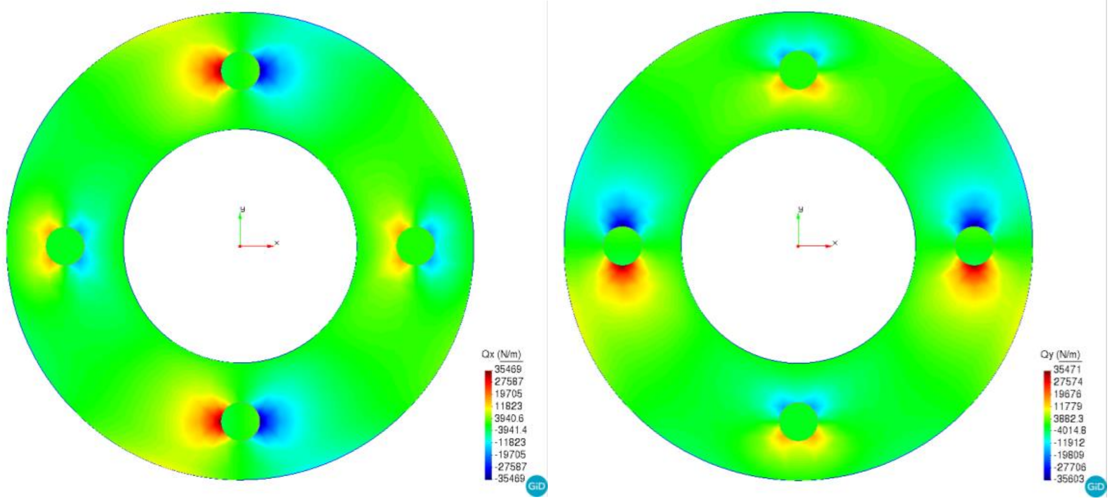


Figure 6. Stresses due to uniform load

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