UPC - BARCELONA TECH
MSc Computational Mechanics
Spring 2018

Computational Structural Mechanics and Dynamics

GiD Homework 3

Prasad ADHAV

Contents

1 Clamped plate with a uniform load

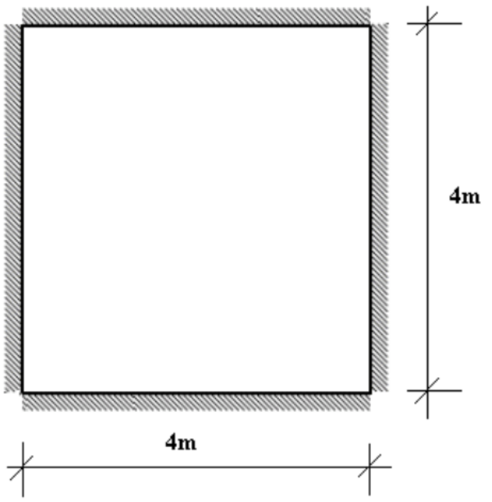


Figure 1.1: Exercise 1: clampled plate with uniform load.

1.1 Purpose of the example

In this first exercise we aim to analyze the state of stress of the plate in Figure 1.1 . The plate is completely clamped and a uniform distributed load is applied. We will use triangular DKT elements, triangular RM elements and quadrilateral CLLL elements to lately perform a comparison with the analytical solution.

1.2 Analysis

1.2.1 Preprocessing

Geometry

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

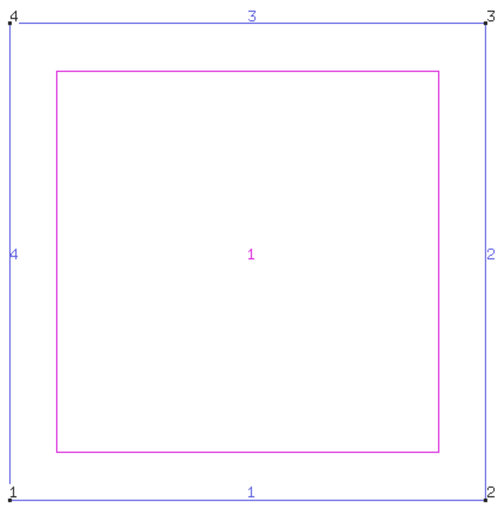


Figure 1.2: Geometry of the plate.

Data

Problem type

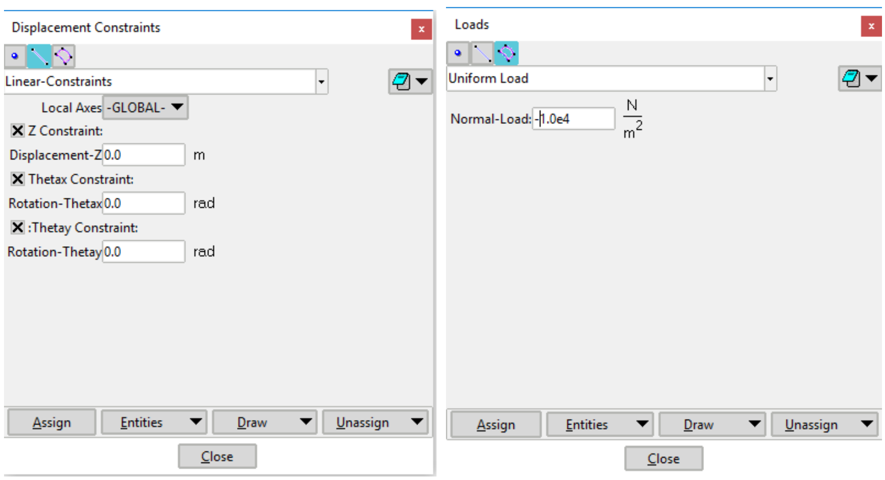
Once the geometry is defined, we can now choose the type of problem that must be solved using *RamSeries*. For this case, we are dealing with plates. Thus, we choose

$$Data/ProblemType/Ramseries\_Educational\_2D/Plates$$

Boundary Conditions

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

- Displacement Constraints / Linear-Constraints: for lines 1, 2, 3 and 4 in Figure 1.2, we assign zero displacements and rotations to simulate the clamped condition.
- Load / Uniform Load : a uniform distributed load of value $q = 1.0e4\ N/m^2$ is considered acting uniformly on the whole plate. We need to put a minus sign in the definition of the load, to ensure that the load is compressive.



(a) (b)

Figure 1.3: Definition of the linear constraint and loading.

Material

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

$$E = 3.0e10\ N/m^2 \quad ; \quad \nu = 0.2 \quad ; \quad t = 0.1\ m$$

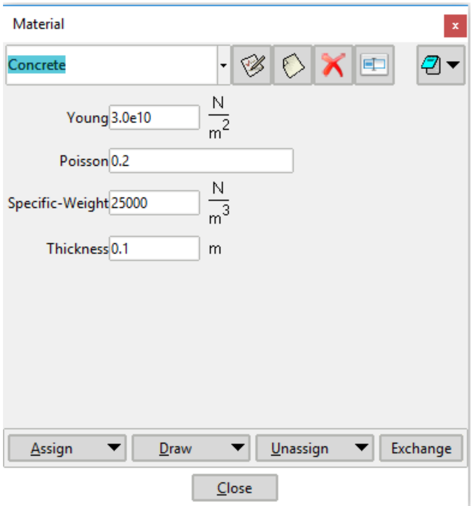


Figure 1.4: Definition of the material of the plate in GiD.

Problem Data

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

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

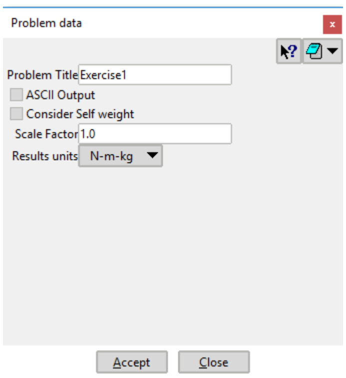


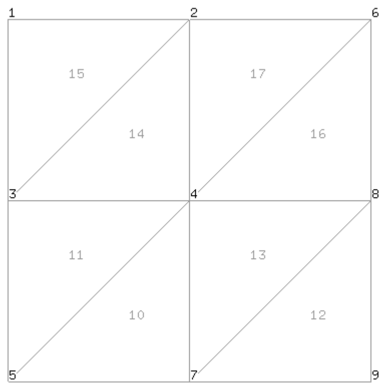
Figure 1.5: Problem data definition for exercise 1.

Mesh

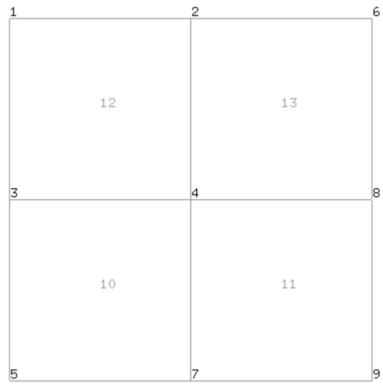
We consider the following meshes for the simulation:

- Structured: we consider a structured mesh.
- Element type: We use a mesh with triangular and quadrilateral elements.
- Element: We consider linear elements with 3 (DKT element) and 4 nodes (CLLL element) and quadratic elements of 6 nodes (RM element.)

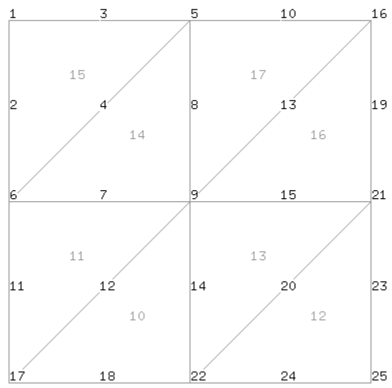
Figure down below is just a comparison of this elements on a coarse mesh.



(a) DKT element (linear triangles)



(b) CLLL element (linear quadrilaterals)



(c) RM element (quadratic triangles)

Figure 1.6: Comparison of the different elements considered on a coarse mesh.

1.2.2 Processing

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

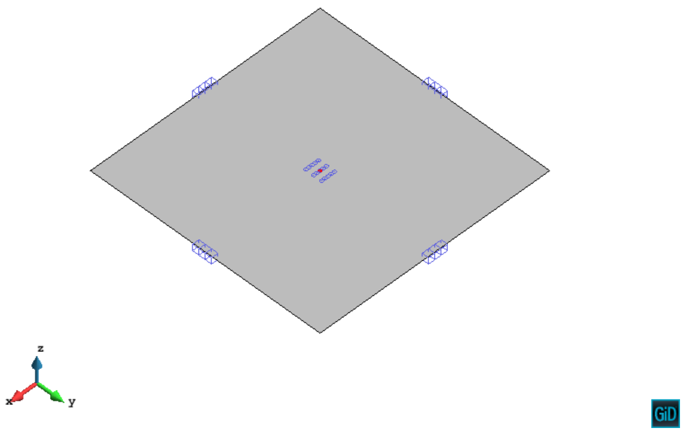


Figure 1.7: Model of the clamped plate with loading

1.2.3 Post-processing and analysis of results

For each element considered in the analysis, we include results for displacements, rotations, bending moments, and shear forces for a particular mesh of element size 0.25. At the end of each subsection, we include a table with the values of the displacement for different meshes, from a coarse one to a finner one, in order to lately compare with the analytical solution at the central node of the plate. The analytical solution is found from the following expression,

$$\delta = \frac{\alpha q L^4}{D} \qquad ; \qquad D = \frac{Et^3}{12(1-\nu^2)} \qquad ; \qquad \alpha = 0.0012657$$

Then, we obtain

$$\delta = \frac{12(1-0.2^2)(0.0012567)(1.0e4)(4^4)}{(3e10)(0.1^3)} = 0.00123538\ m$$

With this information, we can now compute the associated error in the solution.

The following figures show the results obtained from the simulation for the DKT element, the CLLL element and the RM element respectively.

DKT element

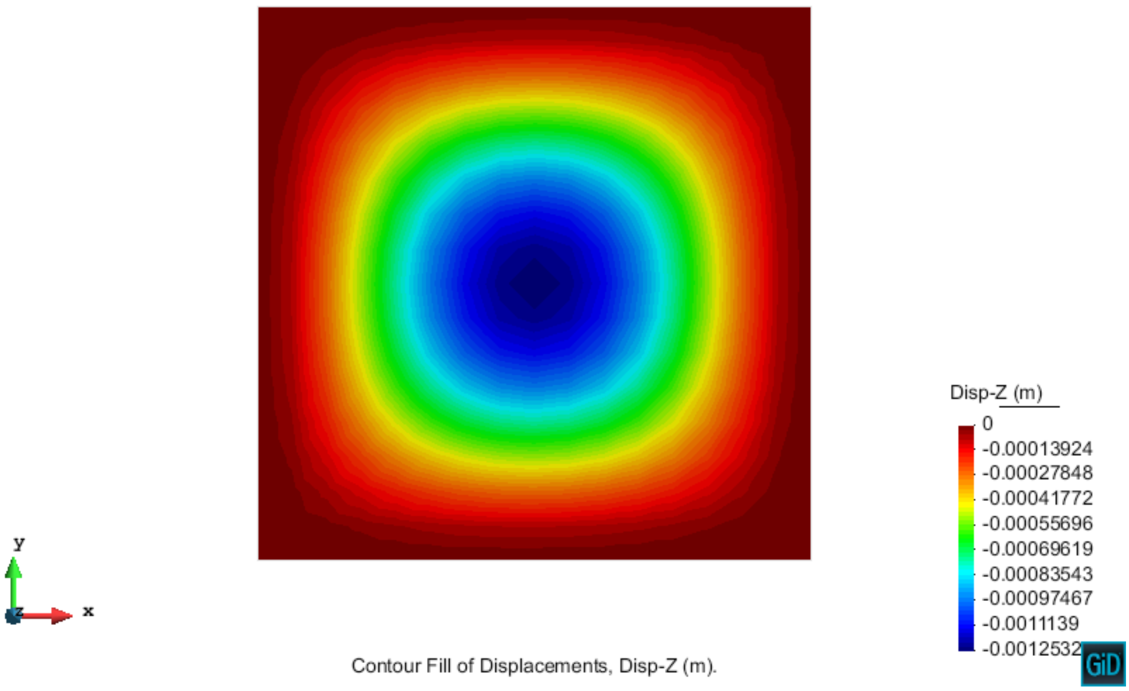


Figure 1.8: Vertical displacements on the plate when analyzed with the DKT element.

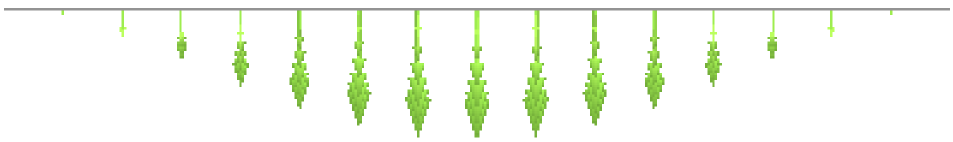


Figure 1.9: Image containing the vector displacements on the plate. As expected, maximum value of displacement is achieved at the center of the plate.

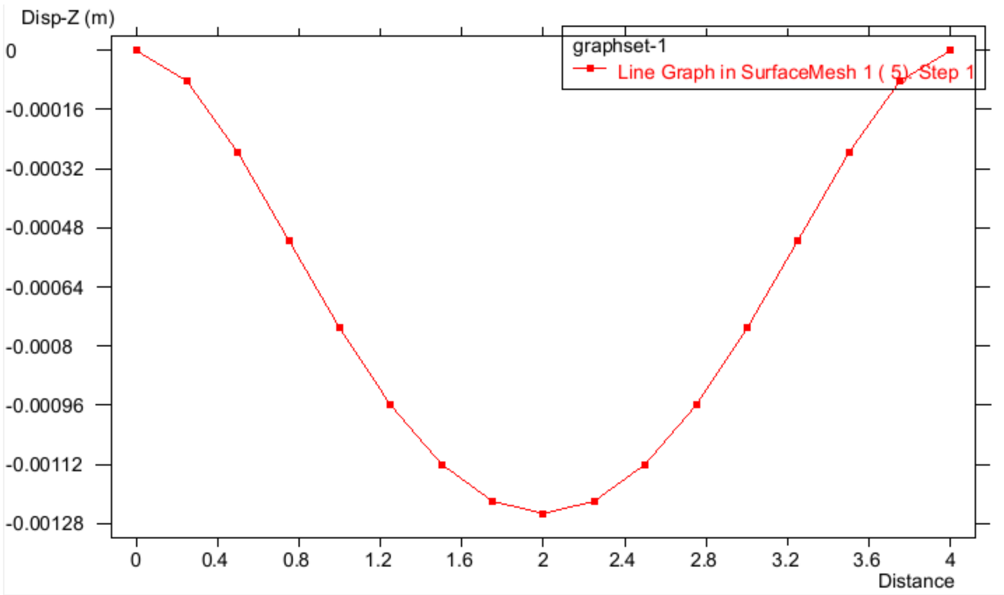


Figure 1.10: Line graph of the displacement along the x direction.

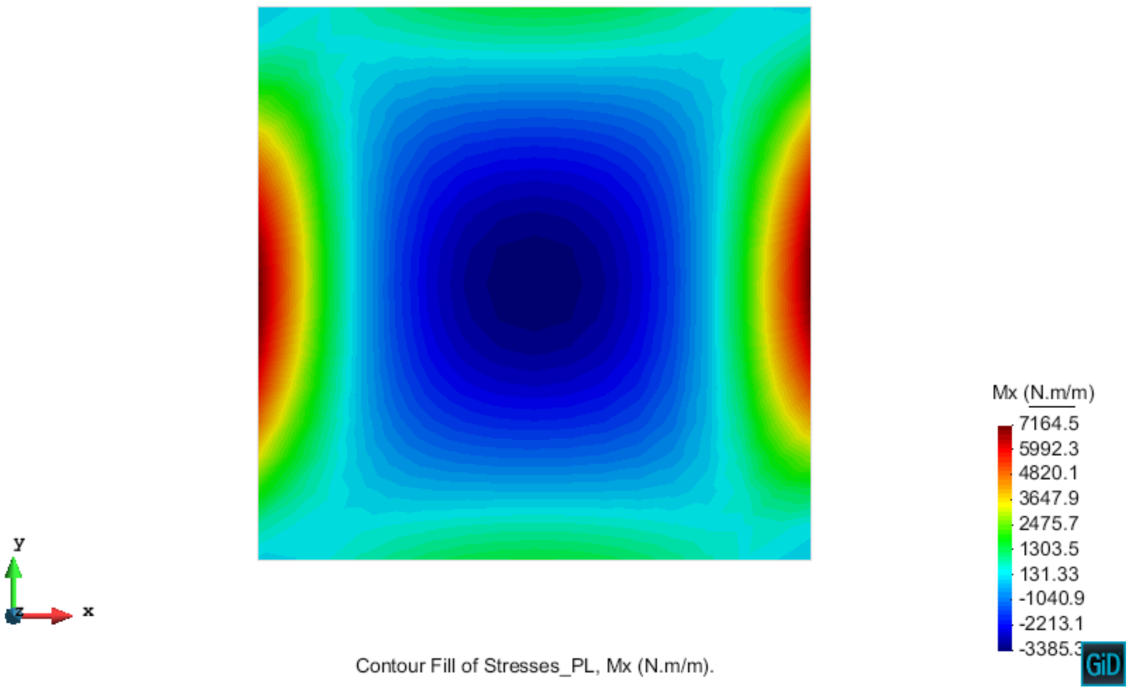


Figure 1.11: Results obtained for the principal bending moment Mx.

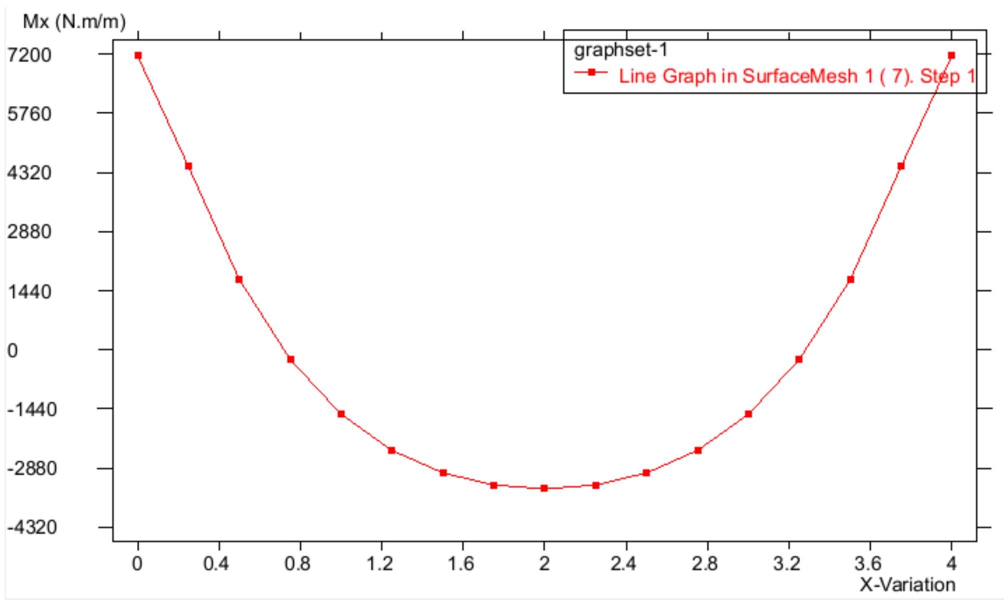


Figure 1.12: Line graph of the principal bending moment M_x along the x axis.

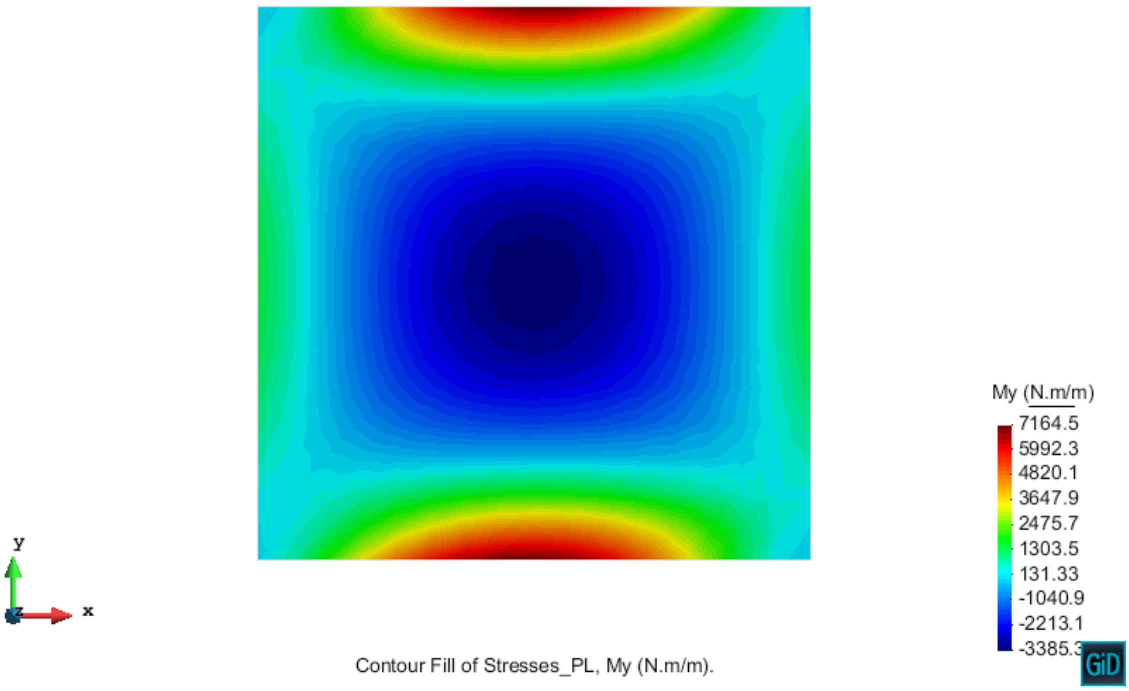


Figure 1.13: Results obtained for the principal bending moment M_y.

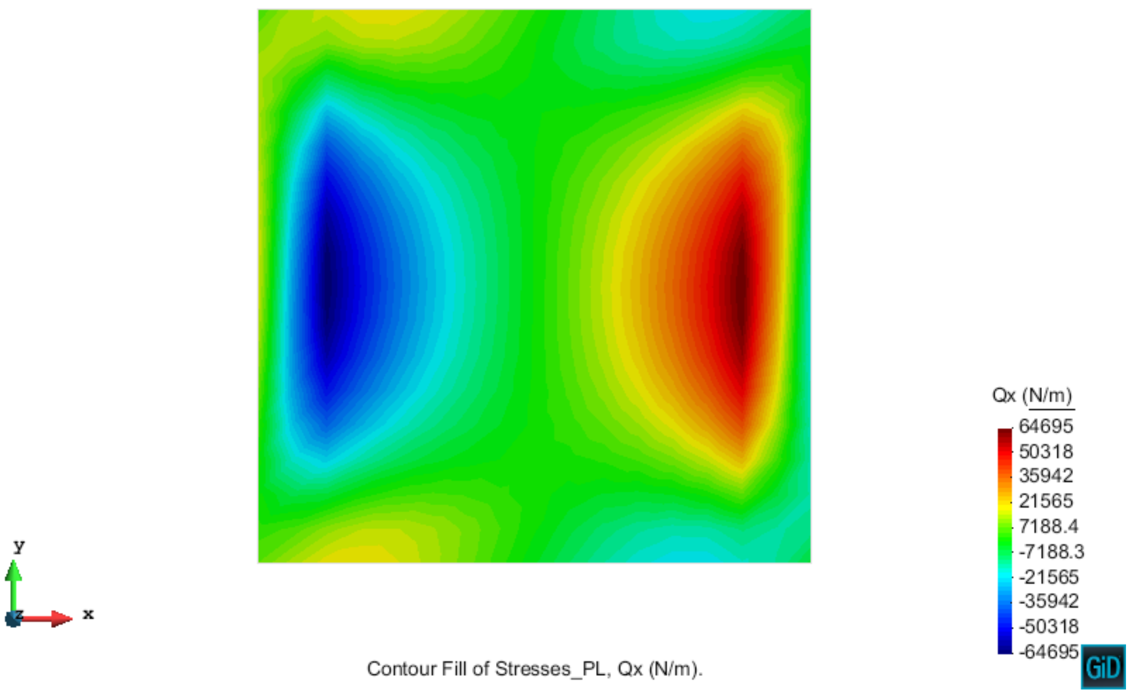


Figure 1.14: Shear force in the x direction, Q_x

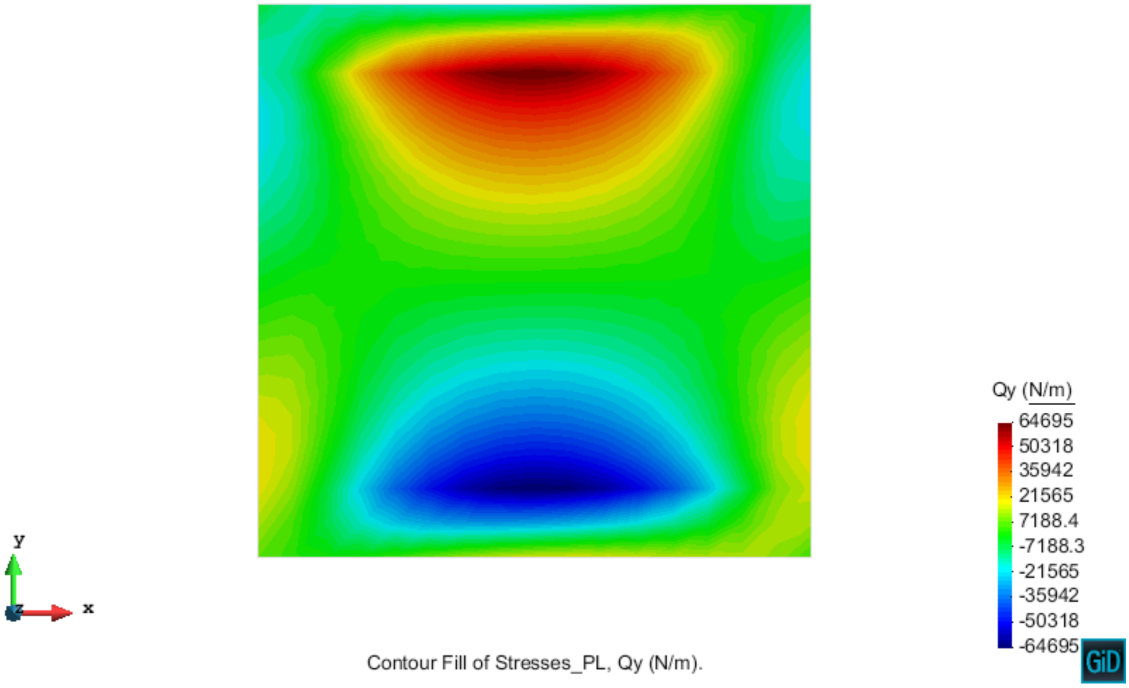


Figure 1.15: Shear force in the y direction, Q_y

Using the results of the analysis with different meshes , the following comparison table was set up for DKT element:

Mesh information				
# Nodes	# Elements	Mesh size	Displacement at central node [m]	Relative error (%)
81	128	0.5	-0.0012796	3.5794
289	512	0.25	-0.0012532	1.4424
1681	3200	0.1	-0.0012454	0.8110
6561	12800	0.05	-0.0012442	0.7139

Table 1.1: Comparison of the result for the central displacement for different meshes using the so-called DKT element.

CLLL element

Figure 1.16: Vertical displacements on the plate when analyzed with the CLLL quadrilateral element.

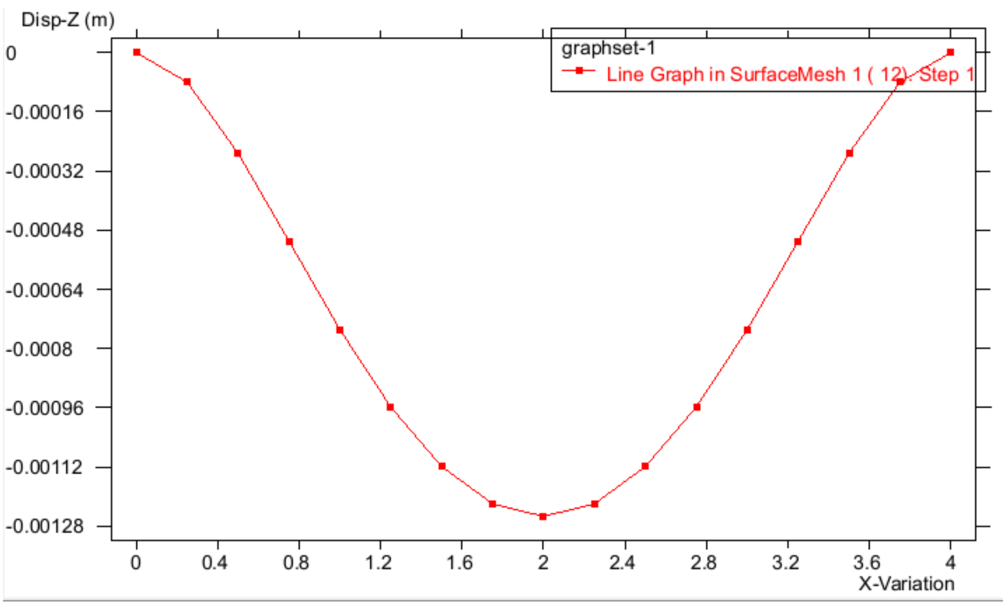


Figure 1.17: Line graph of the displacement along the x direction.

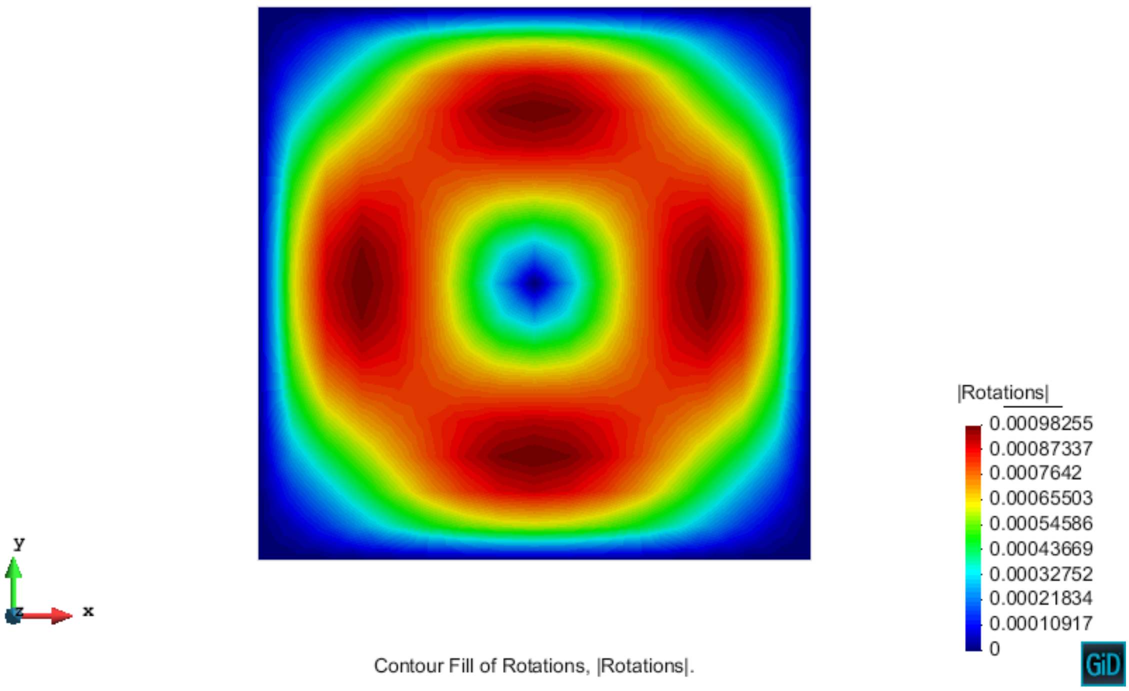


Figure 1.18: Results obtained for the rotations in the whole plate.

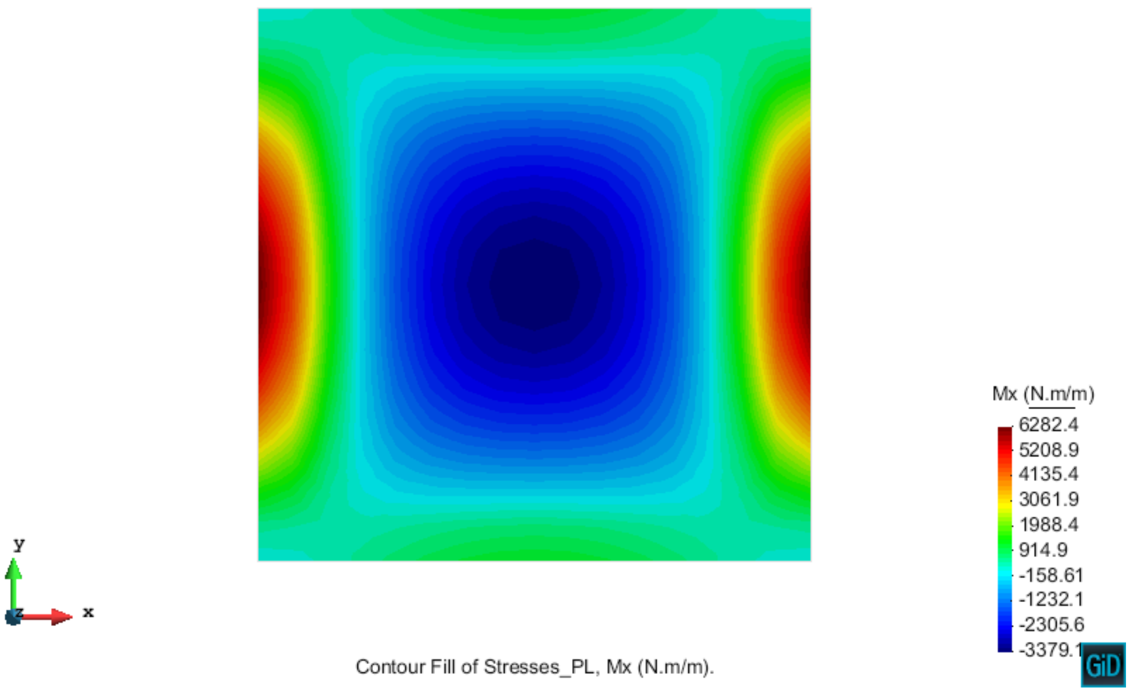


Figure 1.19: Results obtained for the principal bending moment M_x.

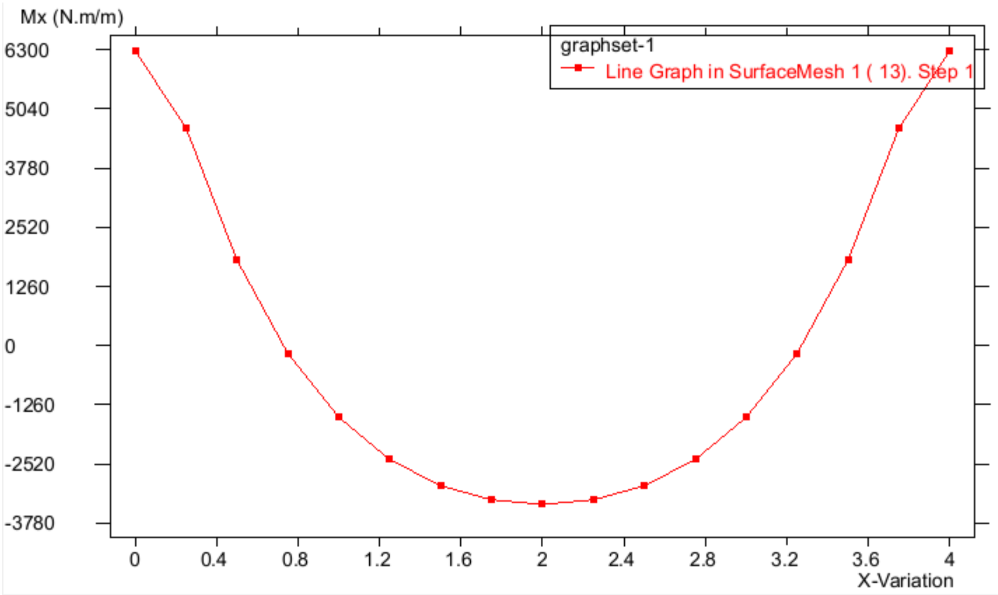


Figure 1.20: Line graph of the principal bending moment M_x along the x axis.

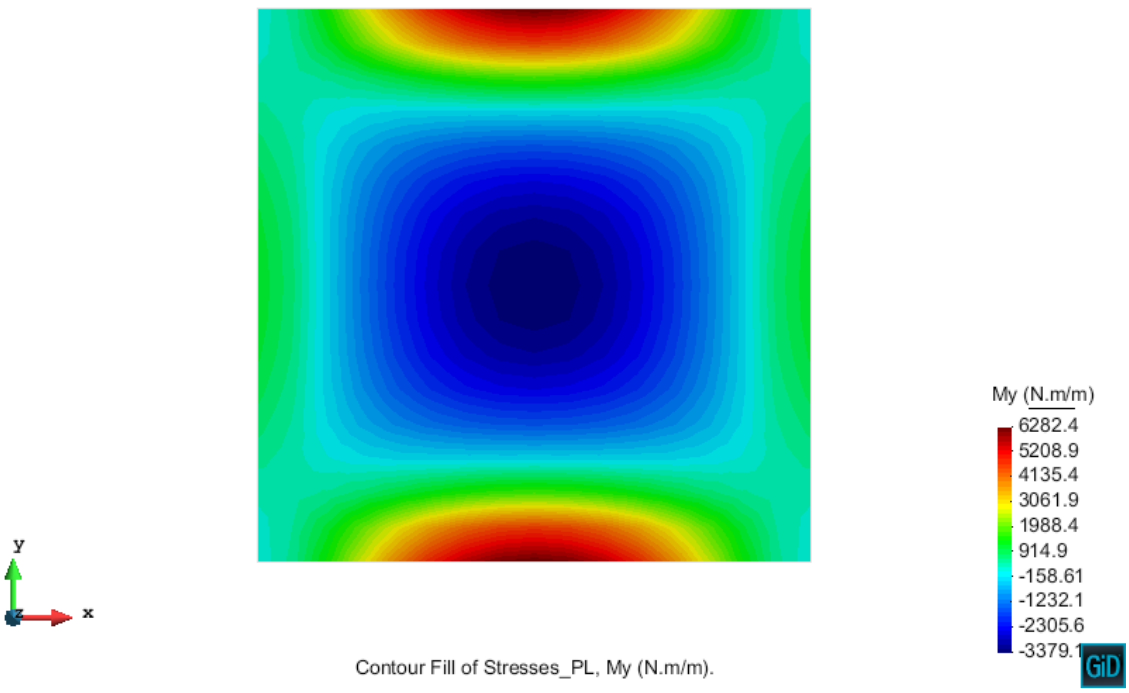


Figure 1.21: Results obtained for the principal bending moment M_y.

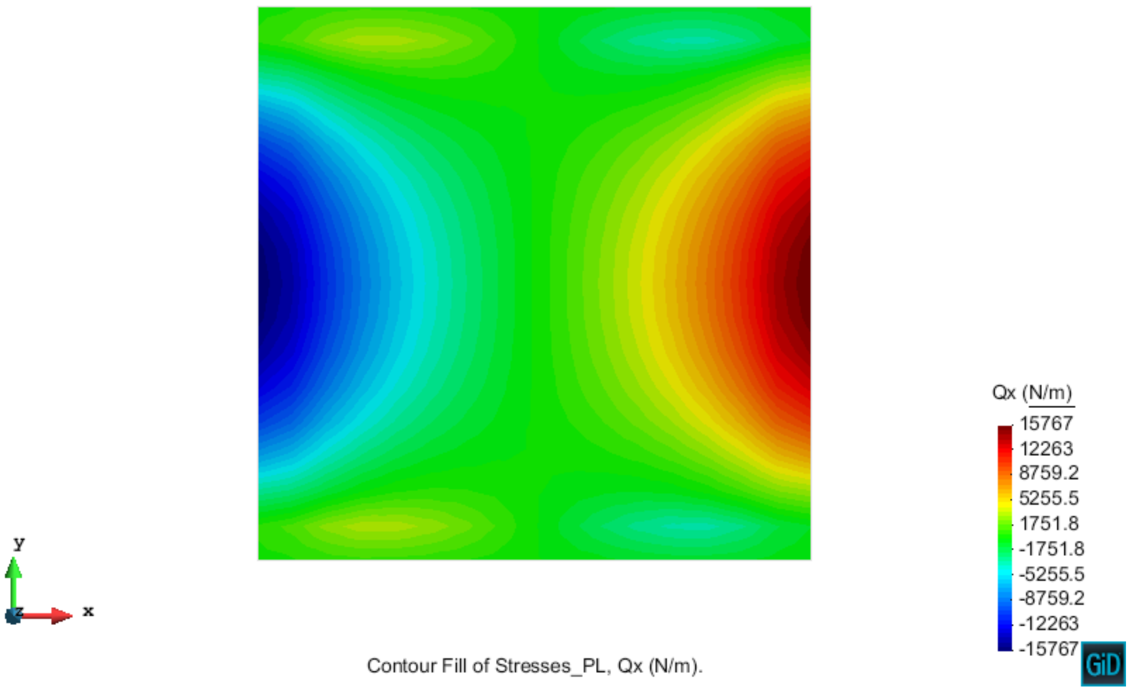


Figure 1.22: Shear force in the x direction, Q_x

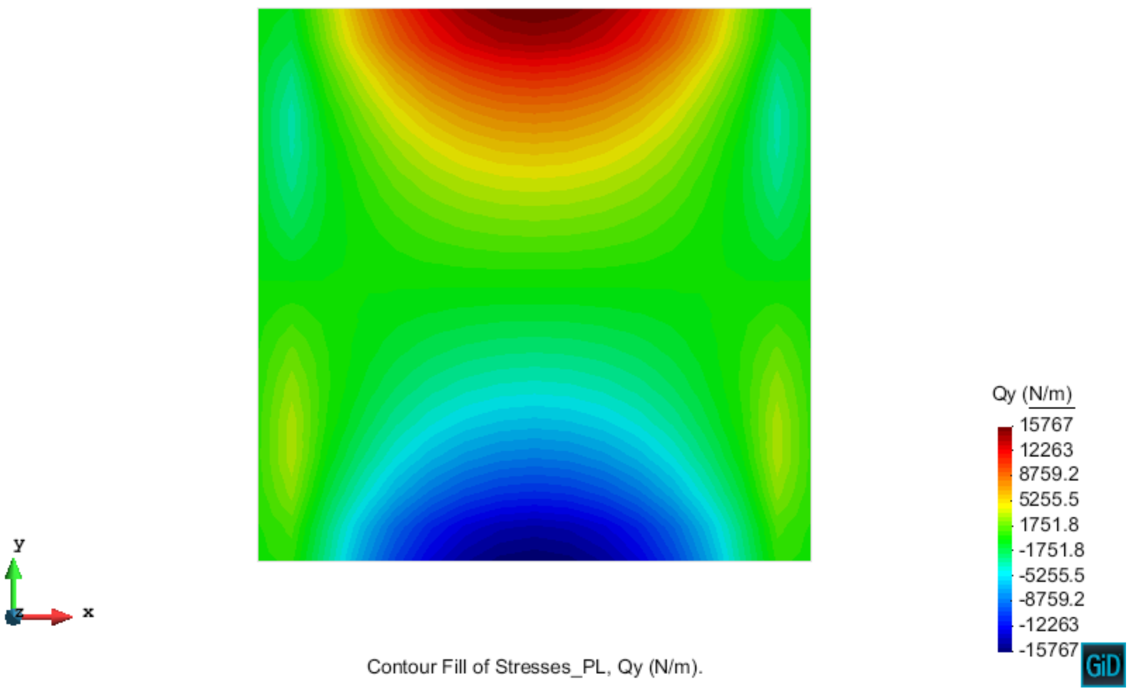


Figure 1.23: Shear force in the y direction, Q_y

Using the results of the analysis with different meshes , the following comparison table was set up for CLLL element:

Mesh information				
nodes	elements	mesh size	displacements	Relative error (%)
81	64	0.5	-0.0012419	0.52777
289	256	0.25	-0.0012534	1.4586
1681	1600	0.1	-0.0012567	1.7257
6561	6400	0.05	-0.0012572	1.7662

Table 1.2: Comparison of the result for the central displacement for different meshes using the so-called CLLL element.

RM element

Figure 1.24: Vertical displacements on the plate when analyzed with the RM quadrilateral element.

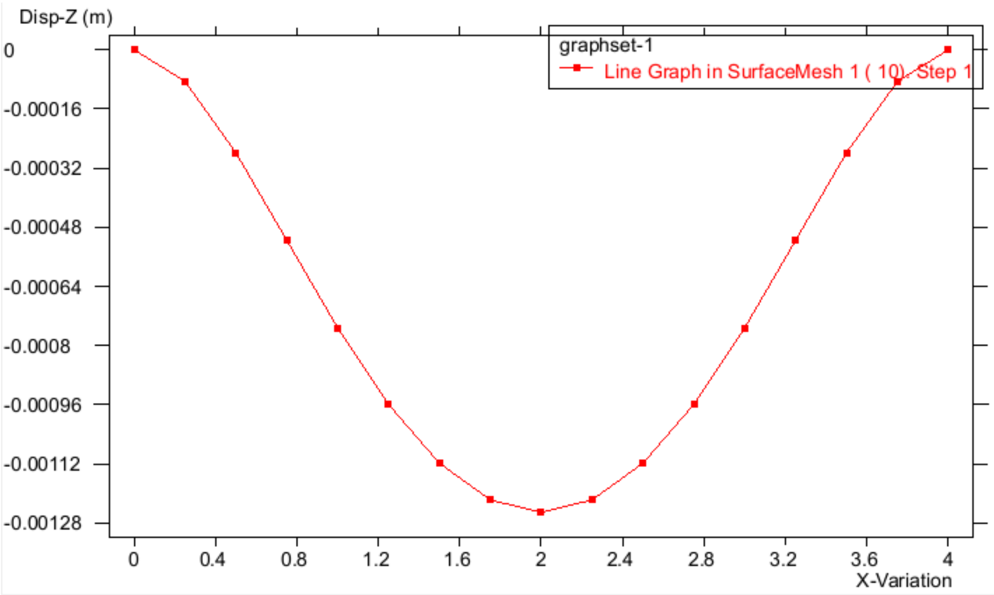


Figure 1.25: Line graph of the displacement along the x direction.

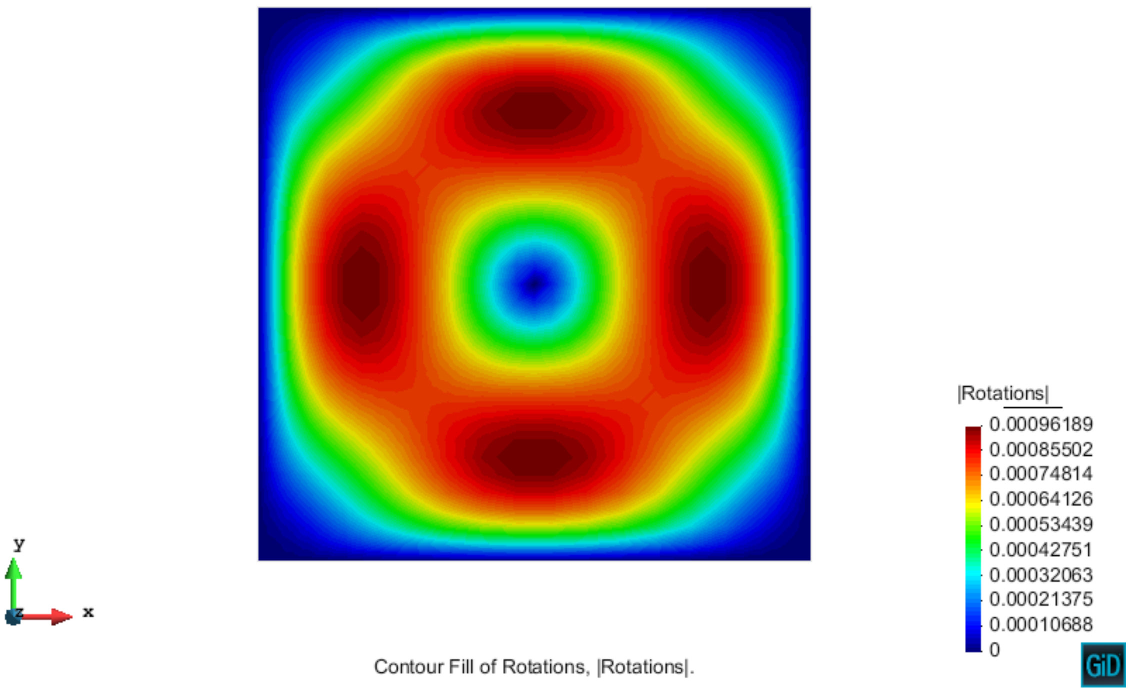


Figure 1.26: Results obtained for the rotations in the whole plate.

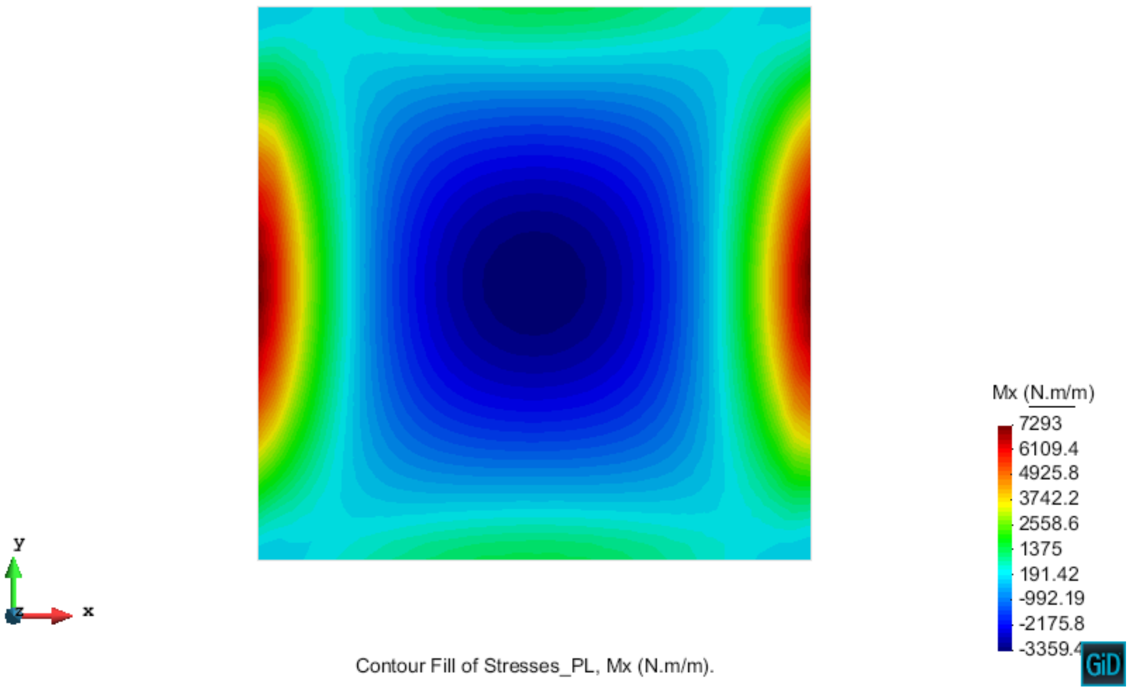


Figure 1.27: Results obtained for the principal bending moment M_x.

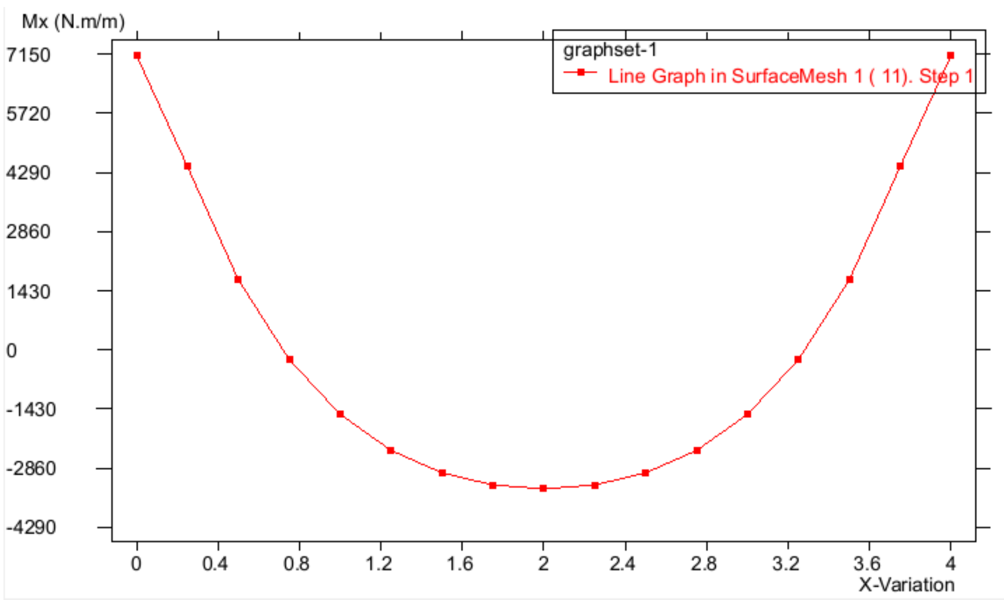


Figure 1.28: Line graph of the principal bending moment M_x along the x axis.

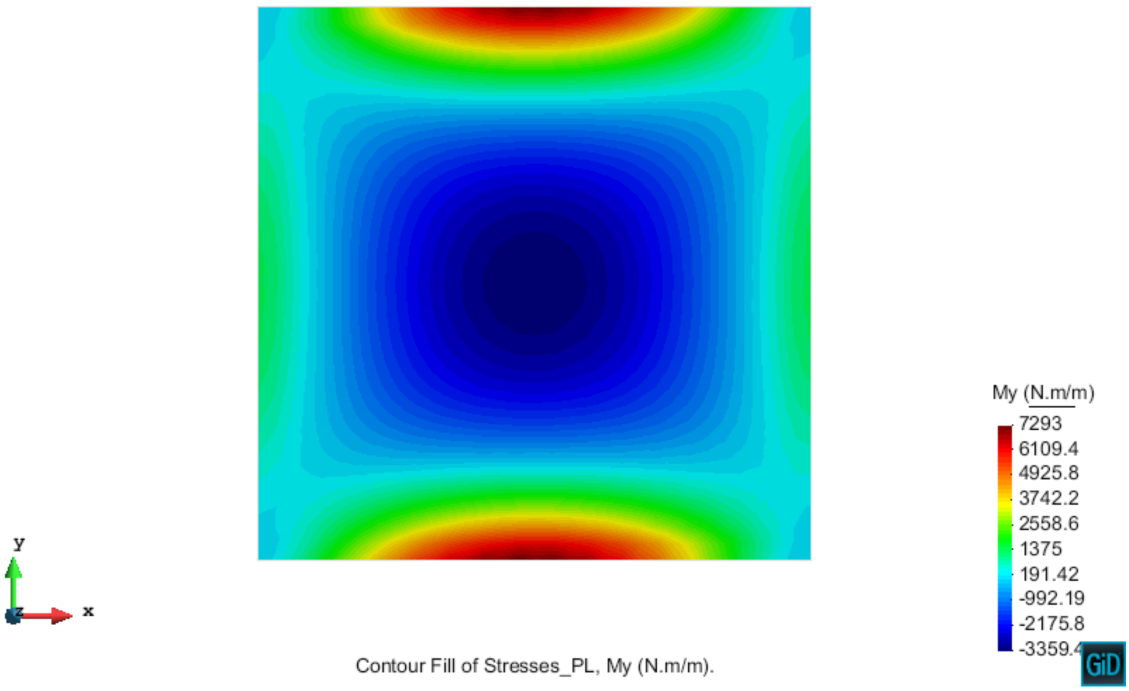


Figure 1.29: Results obtained for the principal bending moment M_y.

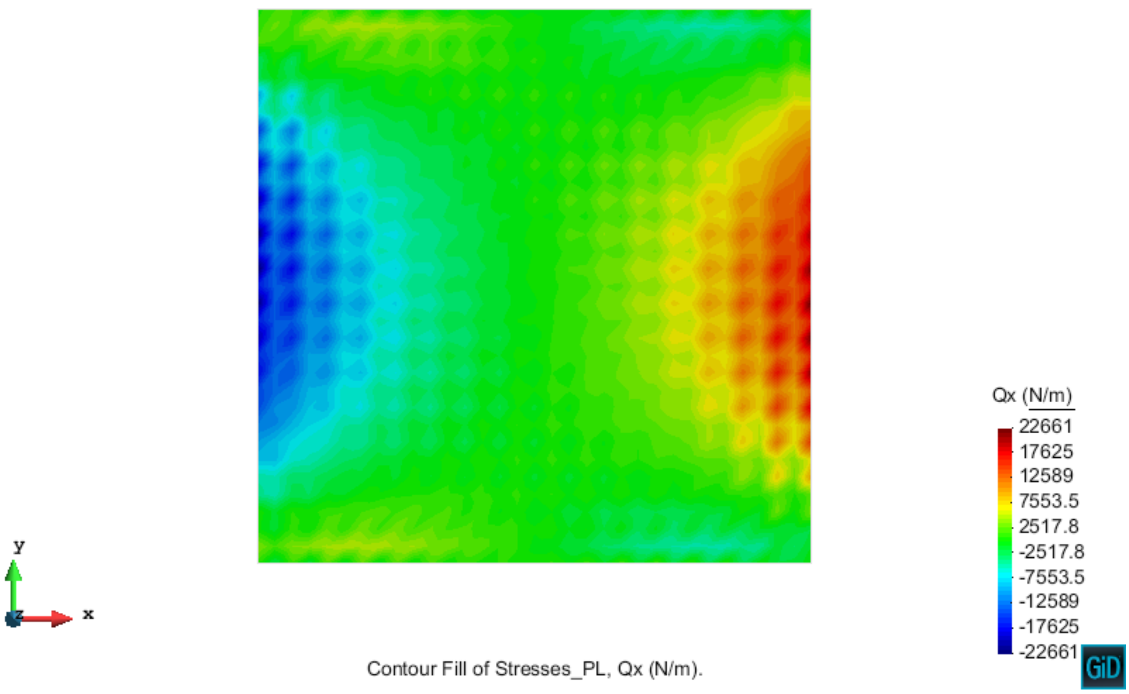


Figure 1.30: Shear force in the x direction, Q_x

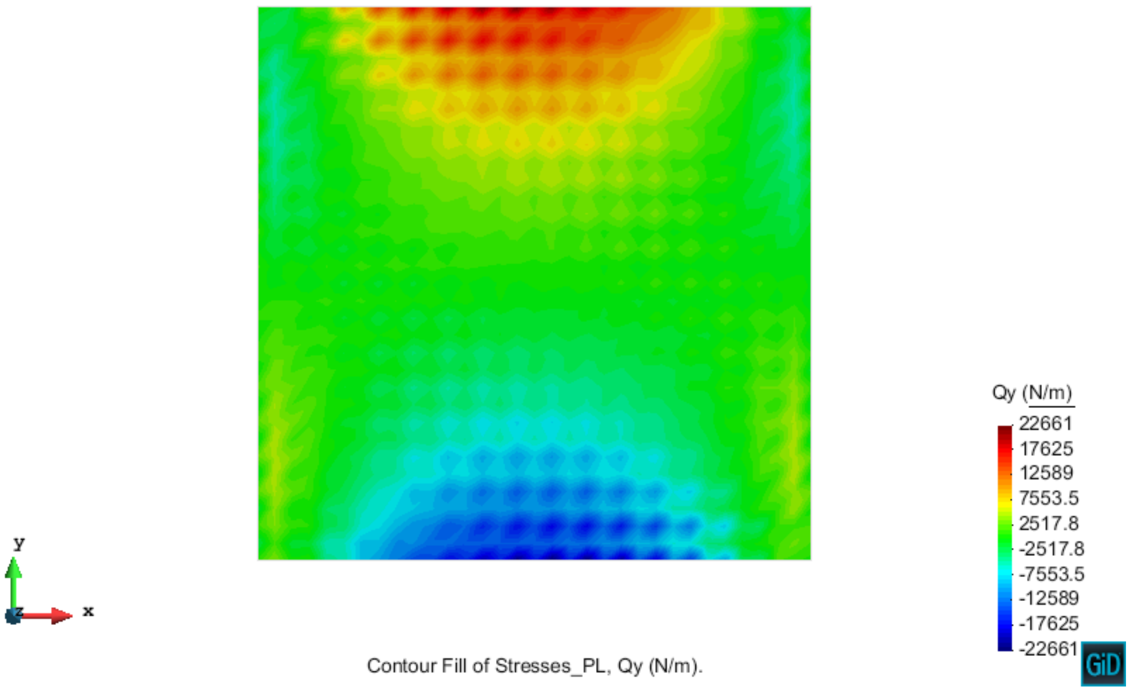


Figure 1.31: Shear force in the y direction, Q_y

Using the results of the analysis with different meshes , the following comparison table was set up for RM element:

Mesh information				
nodes	elements	mesh size	displacements	Relative error (%)
289	128	0.5	-0.0011846	4.1104
1089	512	0.25	-0.0012511	1.2724
6561	3200	0.1	-0.0012572	1.7662
25921	12800	0.05	-0.0012574	1.7824

Table 1.3: Comparison of the result for the central displacement for different meshes using the so-called RM element.

Comments on state of stress

Even though there are some differences in the results depending on the element chosen for the analysis, we can still make a general discussion regarding the solution obtained. As expected for this simple plate completely clamped on the sides under a uniform distributed load, the maximum displacement is achieved in the plate sides, Figures 1.8, 1.16 and 1.24. Similar results for rotations, where the rotation of the central node is zero, as analytical results claim. The solution for the bending moments and shear provide a sort of interpretation of the stress state. It is important to note the complete symmetry of these results (actually same values within an element), as the plate is symmetric and it is symmetrically loaded. We observe that points closer to the center of the geometry suffer maximum compressive bending, Figures 1.11, 1.13, 1.27, 1.29, 1.19 and 1.21. This can also be checked in the corresponding line graphs for the moments. On the other hand, maximum tension bending are produced in points far from the center, but somewhat aligned with the axis of symmetry of the figure. For the case of shear, we see that both maximum positive and negative values are on the extremes of the corresponding axis, e.g. for Q_x we see that the maximum values are on the extremes of the x axis. Figures 1.14, 1.15, 1.22, 1.23, 1.30 and 1.31 give a clear explanation of this fact.

Comments on displacement at central point

With all the information collected on the tables for each element, the next convergence plot is obtained.

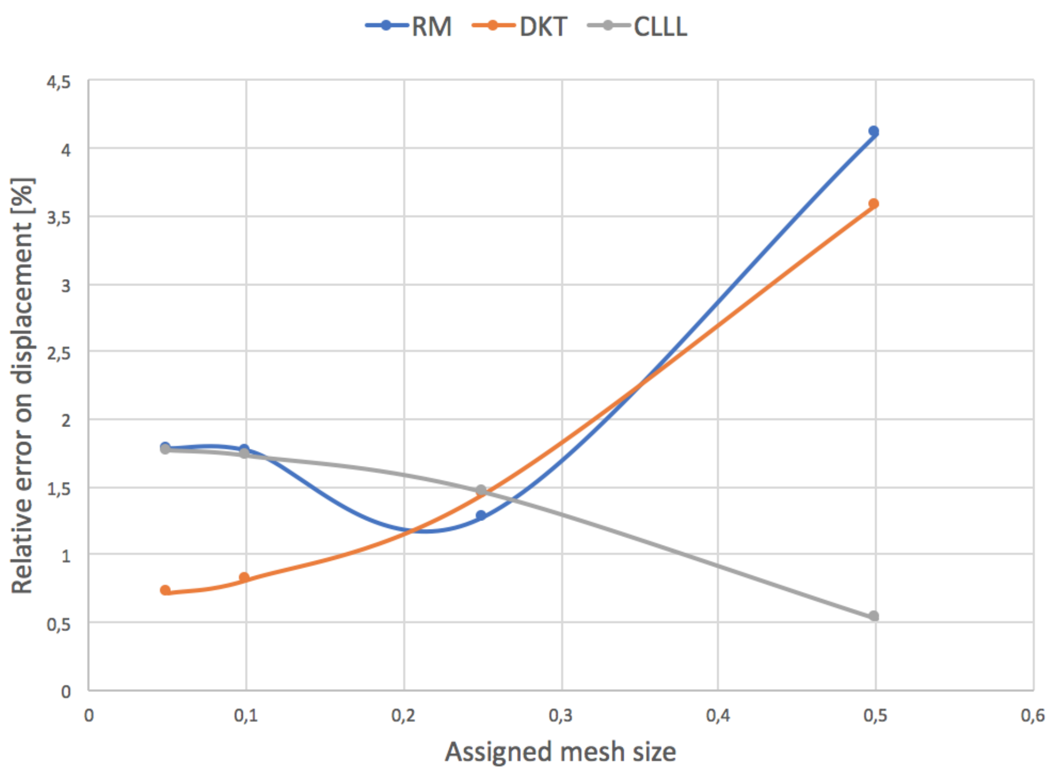
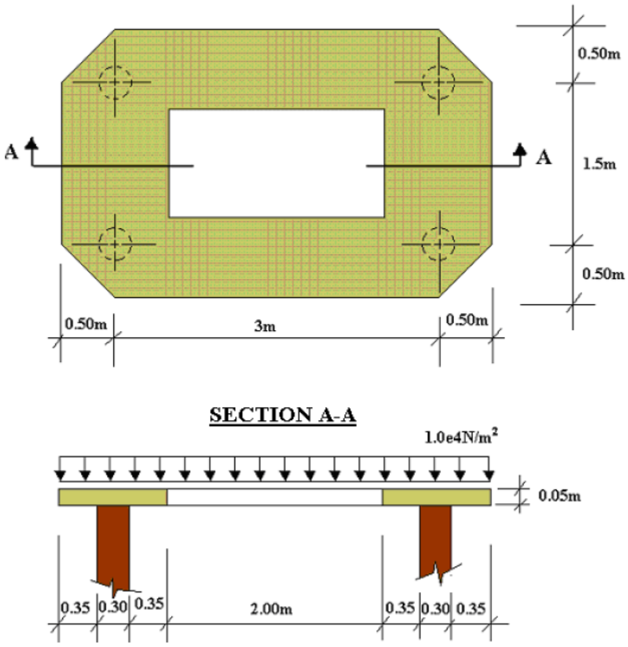


Figure 1.32: Convergence plot for the different elements considered. Relative error with the considered exact solution is considered in the y axis and the assigned mesh size in the x axis.

We observe that the quadratic element (RM element) has the fastest rate of convergence for coarser mesh sizes (between 0.3 and 0.5), but when getting closer to the solution it starts to oscillate a little bit and we got two diverging results but still pretty accurate. The case of the CLLL element is somewhat curios because it appears to increase its error as we approach analytical solution, even though we assume that for more finner meshes, the results start to converge [1]. An important fact is that for the coarsest mesh considered in the analysis, it is the element which provides the mos accurate result. Finally, the DKT (linear triangle) element is the one which really appears to strictly converge to the computed analytical solution. As one can see in [1], the DKT element performs really well for this type of problems. It is also important to note that the analytical solution, is a result of the truncation of a Fourier series with the coefficient α. Thus, this can be also the explanation for some of the diverge issues found in the analysis.

[1] The finest mesh we were able to work with was one with mesh size of 0.025. Finer meshes came up with an error in GiD.

2 Thin plate with internal hole.



2.1 Purpose of the example

In this exercise the goal is to analyze the structural behaviour of a plate using the thin plate theory.

2.2 Analysis

2.2.1 Preprocessing

Geometry

First, the geometry is created using the GiD sketcher tool.

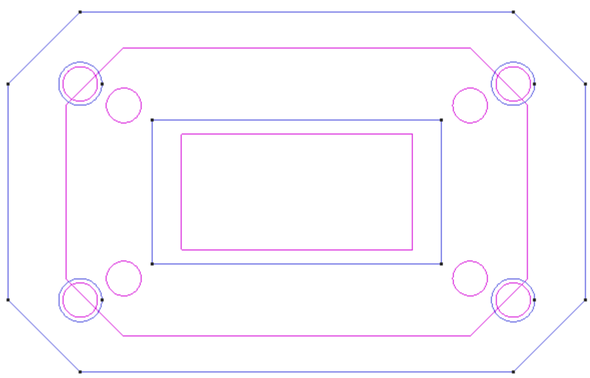


Figure 2.1: Geometry of the plate

Data

Problem type

Once the geometry is defined, the type of problem that must be solved should be chosen using *RamSeries* solver. For this case, the plate theory should be applied. Thus, using GiD's menu:

$$Data/ProblemType/Ramseries\_Educational\_2D/Plates$$

Boundary Conditions

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

- Elastic Constraints / Linear Elast.-Constraints: an elastic constraint with a very high value of K (similar to the Young modulus of concrete) was applied to the contact area between the cylindrical supports and the plate in the Z direction. This allows to properly simulate the condition in those points: they can move but their displacement is highly restricted.

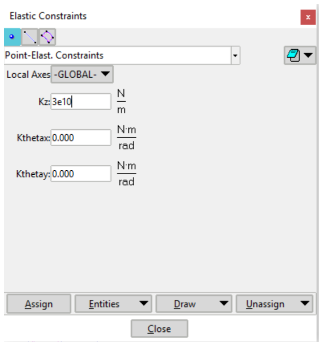


Figure 2.2: Definition of the elastic constraints.

- Load / Uniform Load : a uniform distributed load is applied over all the surface of the plate.

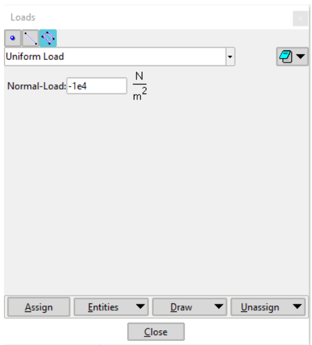


Figure 2.3: Definition of the uniform load over the plate

Material

The plate is made of steel with the following mechanical characteristics:

$$E = 2.1e11 \quad N/m^2 \qquad ; \qquad \nu = 0.3 \qquad ; \qquad \gamma = 7.80e4 \quad N/m^3$$

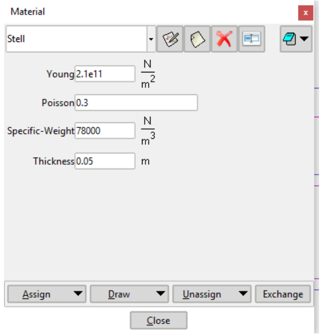


Figure 2.4: Material of the plate on GiD.

Problem Data

In this section some additional data required for the analysis is specified.

- Problem title: Prac3_Ex2
- ASCII Output: NO
- Consider self weight: No
- Scale factor: 1.0
- Result Units: N-m-kg

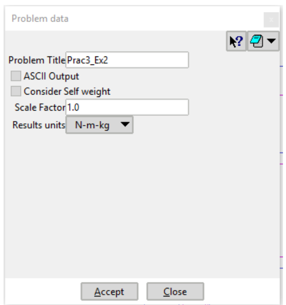


Figure 2.5: Problem data definition for exercise 2.

Mesh

The following mesh is defined for the simulation:

- Unstructured: non special requirements are set on the statement of the problem about the structure.
- Element type: triangular
- Linear element: linear DKT triangles

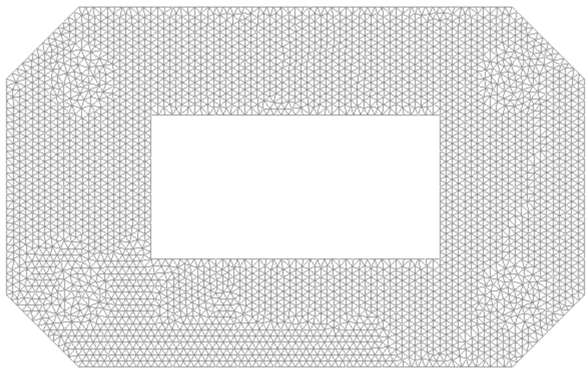


Figure 2.6: Mesh of DKT triangular elements for the simulation of exercise 2.

2.2.2 Processing

Once the mesh is generated, the problem can be sent to be calculated.

2.2.3 Postprocessing and analysis of results

The following figures show the results of the simulation for the displacement. The structure's displacemente is shown both as a contour field and as vectors to facilitate the visualization.

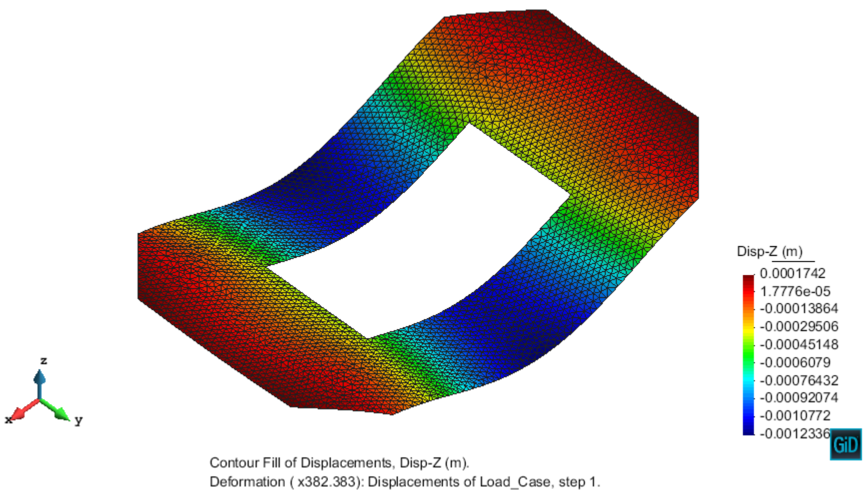


Figure 2.7: Isometric view of displacements in the Z direction over deformed shape.

As it can be expected, the plate bends at the middle while the edges remain almost flat. Its maximum deflection is located at the middle point of the length (X direction) and there is almost zero displacement in the edges were the supports are located, as it can be seen in Figure 2.8. The deformation is uniform in the Y direction, as can be easily appreciated at the contour plot shown in Figure 2.7.

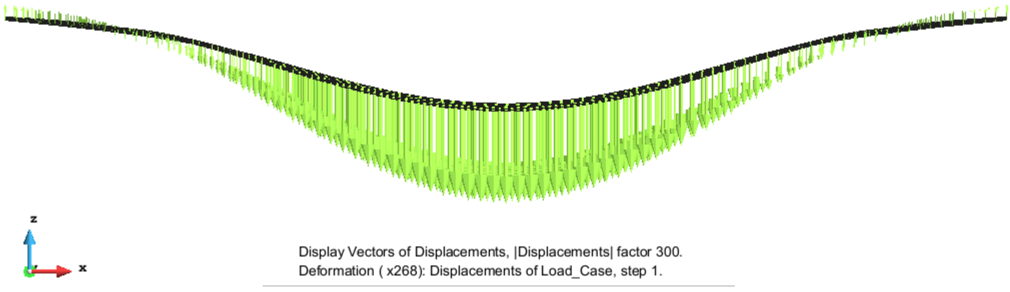


Figure 2.8: Isometric view of displacements in the Z direction over deformed shape.

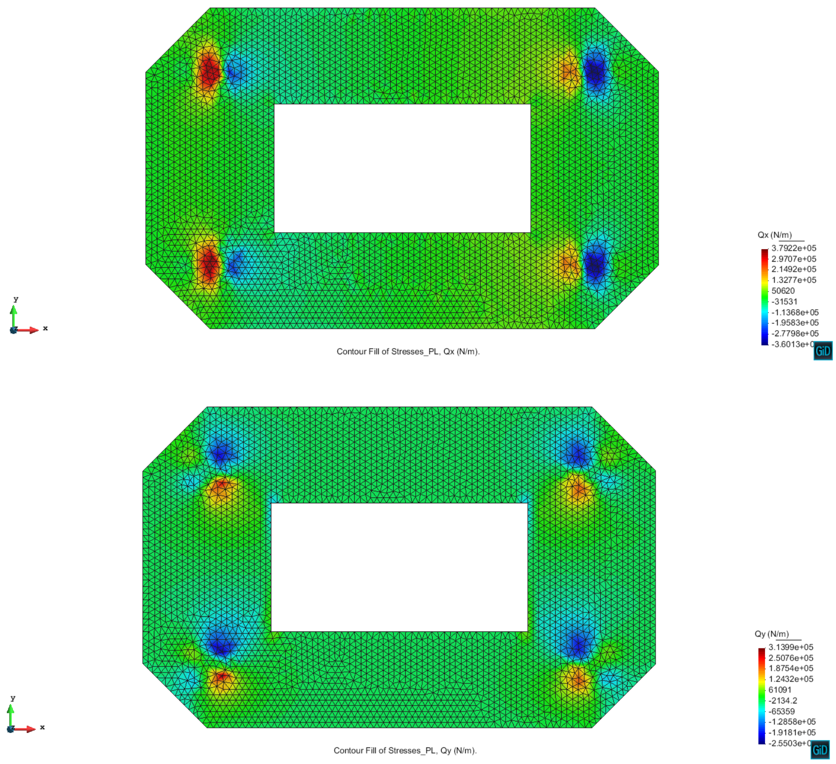


Figure 2.9: Stresses due to the applied uniform load

Figure 2.9 shows the stress distribution over the plate due to the applied load. The stress field is almost uniform all over the surface with certain concentration were the supports are located. The distribution obtain is symmetric, if the scale is carefully observed it is possible to notice that the maximum and minimum stresses are almost the same in absolute value.

The structure could have been reduced to only a quarter of it using symmetry and applying the appropriate boundary conditions, which would allow to reduce the computational cost. Also, the mesh could be modify in order to concentrate more nodes around the areas where the supports are located.

For the results shown before and as indicated in the 'Problem data' section, the weight of the structure was not considered. A simulation taking into account the contribution of the dead weight of the plate was run and the displacements obtained are shown in Figure 2.10. As it could be anticipated, the distribution of the field is the same as before with the difference that now the displacements are higher. This effect is perfectly understandable, as the dead weight could be modeled as a distributed force acting on top of the plate, the same as the external applied load.

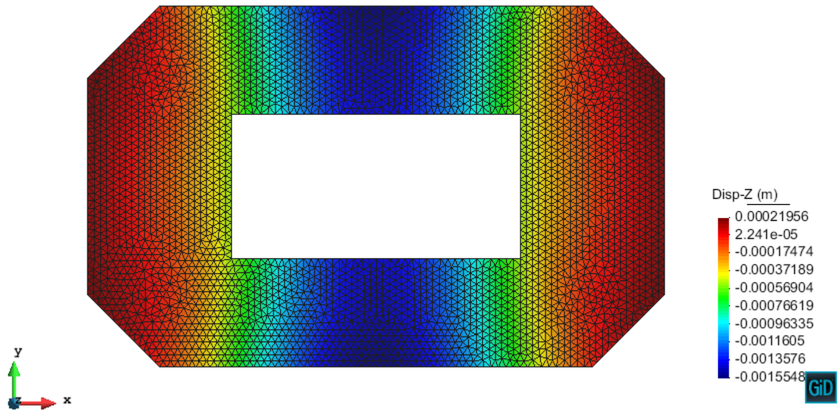
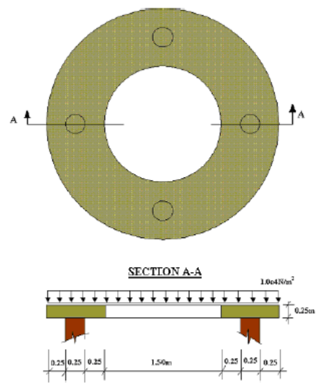


Figure 2.10: Displacements in the Z direction over the plate considering dead weight.

3 Thick circular plate with internal hole.



3.1 Purpose of the example

In this exercise the goal is to analyze the structural behavior of a reinforced concrete circular plate using the thick plate theory.

3.2 Analysis

3.2.1 Preprocessing

Geometry

First, the geometry is created using the GiD sketcher tool.

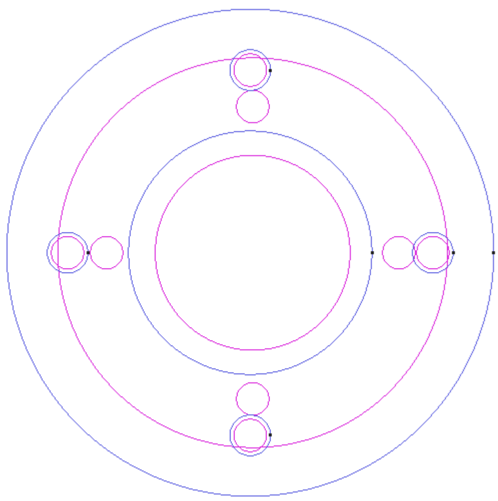


Figure 3.1: Geometry of the circular plate

Data

Problem type

Once the geometry is defined, the type of problem that must be solved should be chosen using *RamSeries* solver. For this case, the plate theory should be applied. Thus, using GiD's menu:

Data/ProblemType/Ramseries_Educational_2D/Plates

Boundary Conditions

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

- Elastic Constraints / Linear Elast.-Constraints: an elastic constraint with a very high value of K (similar to the Young modulus of concrete) was applied to the contact area between the cylindrical supports and the plate in the Z direction. This allows to properly simulate the condition in those points: they can move but their displacement is highly restricted.

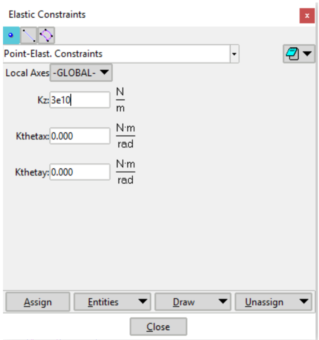


Figure 3.2: Definition of the elastic constraints.

- Load / Uniform Load : a uniform distributed load is applied over all the surface of the plate.

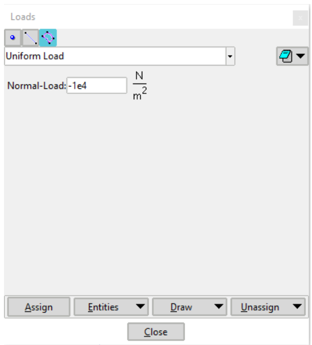


Figure 3.3: Definition of the uniform load over the plate

Material

The plate is made of steel with the following mechanical characteristics:

$$E = 2.1e11 \quad N/m^2 \qquad ; \qquad \nu = 0.3 \qquad ; \qquad \gamma = 7.80e4 \quad N/m^3$$

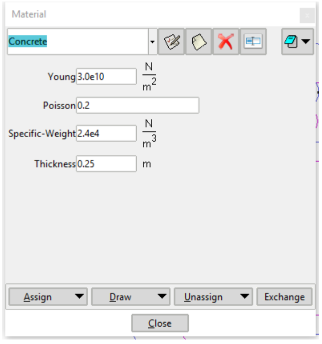


Figure 3.4: Material of the plate on GiD.

Problem Data

In this section some additional data required for the analysis is specified.

- Problem title: Prac3_Ex2
- ASCII Output: NO
- Consider self weight: Yes
- Scale factor: 1.0
- Result Units: N-m-kg

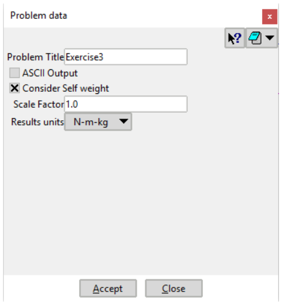


Figure 3.5: Problem data definition for exercise 3.

Mesh

The following mesh is defined for the simulation:

- Unstructured: non special requirements are set on the statement of the problem about the structure.
- Element type: triangular
- Linear element: Reissner-Mindlin six node triangles with reduced integration

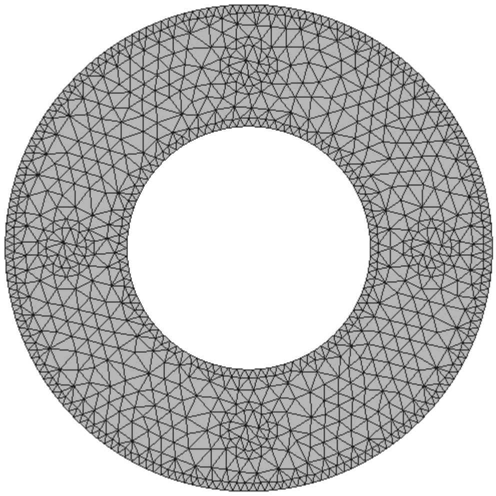


Figure 3.6: Mesh of R-M triangular elements for the simulation of exercise 3.

3.2.2 Processing

Once the mesh is generated, the problem can be sent to be calculated.

3.2.3 Postprocessing and analysis of results

The following figures show the results of the simulation.

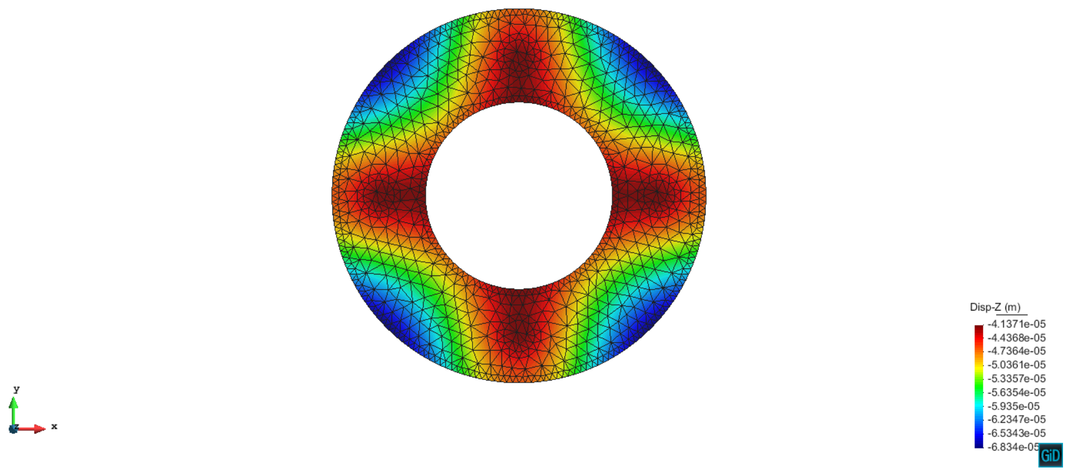


Figure 3.7: Displacements in the Z direction over the plate.

As it could be anticipated and as shown in Figure 3.7, the behavior of the structure is perfectly symmetric. As with the previous case, the problem can be reduced to a quarter of the whole structure and then solved using the appropriate boundary conditions.

In general, it could be said that the displacements of the structure are not very important, as they are all of the order of $10^{-5}m$. This geometry is much more stiff than the one presented in the previous example, because the plate is much thicker and the distance between supports is smaller. The highest displacements are located in the most external edge between the supports, and of course the smallest displacements are located where the supports are. However, the different between the maximum and minimum is not significant, as it can be interpreted from the displacement vectors shown in Figure 3.8.

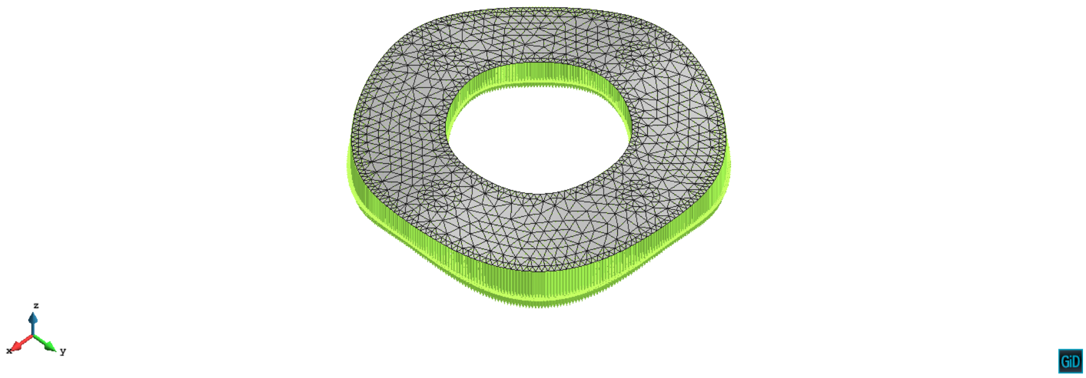


Figure 3.8: Isometric view of displacements in the Z direction over deformed shape.

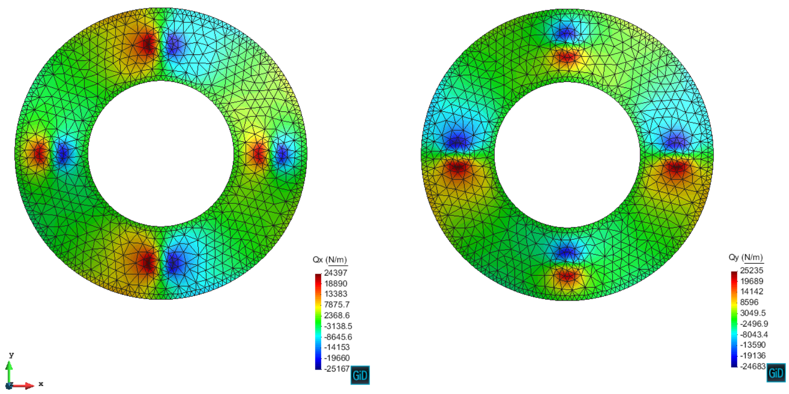


Figure 3.9: Stresses due to the applied uniform load

Figure 3.9 shows the stress distribution over the plate due to the applied load and dead weight. The stress field is quite uniform all over the surface with some concentration were the supports are located. The distribution obtain is symmetric, if the scale is carefully observed it is possible to notice that the maximum and minimum stresses are almost the same in absolute value. The structure is very safe and the values of the stress field are still far from the limits of the concrete.

References

[1] Oñate, E., *Structural Analysis with the Finite Element Method. Linear Statics. Volume 2. Beams, Plates and Shells*, First edition, CIMNE 2013.