Course:
Computational Structural Mechanics and Dynamics

Assignment-8

Student: Marcello Rubino

Exercise 1

Analyze the following concrete hyperbolic Shell under self-weight. Explain the behavior of all the Stresses presented.

1.1 Building the hyperbolic structure

Using an Excel sheet it's been possible to calculate the nodal coordinates of the structure, and knowing the extremal four nodal coordinates of the structure it's been used a linear interpolation. The Excel file has been saved as a text file containing only the X,Y,Z coordinates of all the nodes of the structure. Inside the GiD geometrical interface it's imported the nodal infos and using line (firstly) and surface (secondly) tools the geometry has been completed. Some of the surfaces that have built automatically presented the normal vector oriented "against" the global Z, while others oriented like Z (and this is a wrong scenario for the assembly of the global stiffness matrix), so it's been necessary to rotate the orientation of these surfaces.

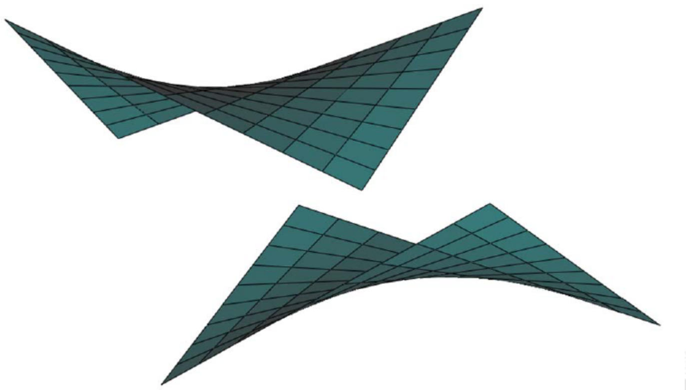


Figure 1a - GiD model of the structure (seen in 3D space)

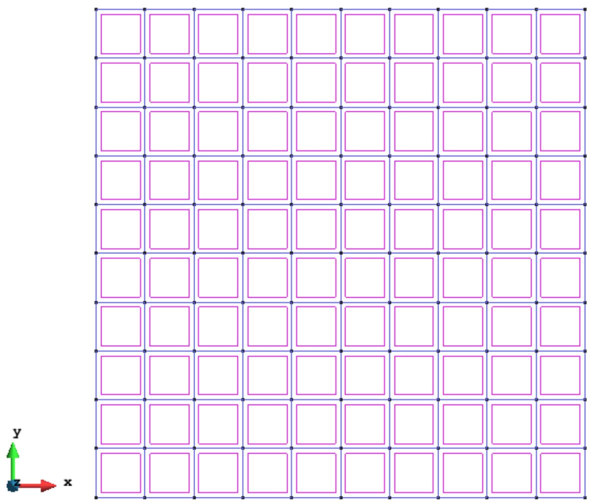


Figure 1b - GiD model of the structure (seen in the XY plane from the top)

1.2 Working on the GiD model

In the GiD project it's been used the "MAT-fem_Shells" tool in order to create an input file for a MatLab code that implements the Reissner-Mindlin structural model ('Lamina_T_RM.m' file).
Firstly, the boundary conditions and material property have been assigned to the structure. Concerning about the B.C., the four sides of the structure have been clamped (using "displacement BC to lines" tool). All 5 DOFs of the structured have been fixed to 0. Then concrete material has been assigned to the surfaces of the model and the thickness fixed = 0.1 m. It's been necessary to "switch-on" the possibility to consider the self-weight for the calculation.

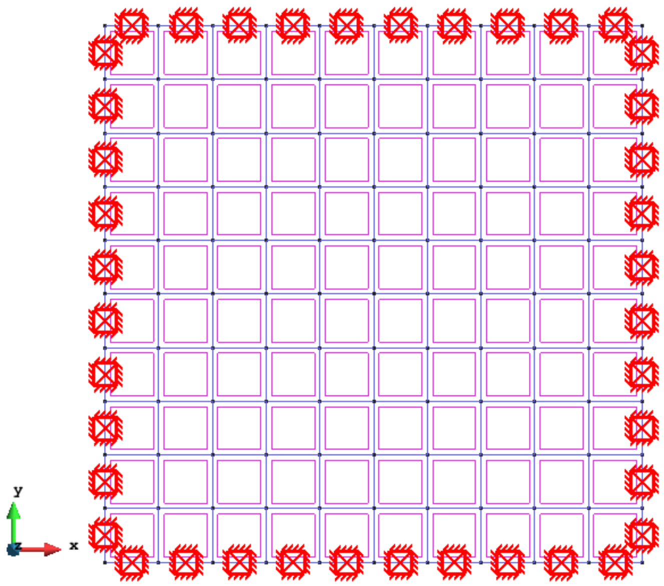
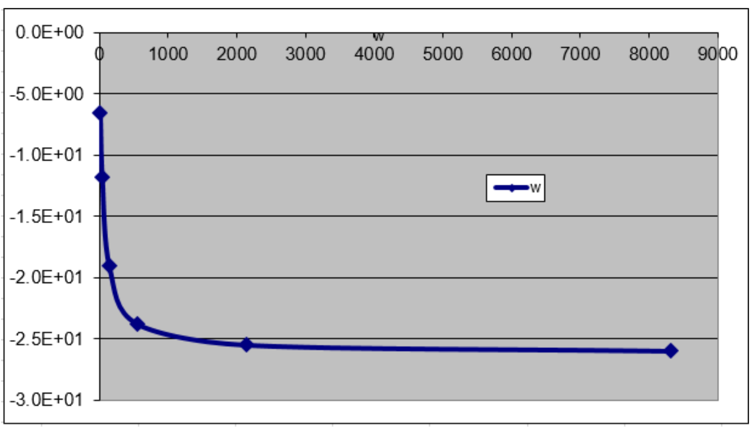


Figure 2 - Boundary Conditions (seen in the XY plane from the top)

Secondly, structured-type mesh has been created in the structure. The type of the elements is triangular and linear, since the MatLab code works with the 3-node triangles. In this case the size of the mesh used is of 0.25 meters, since the size of each surface is of 1 meter and, as shown in the graph 1, reaches a good convergence (3200 elements and 1681 nodes).



Graph 1 - Convergence graph

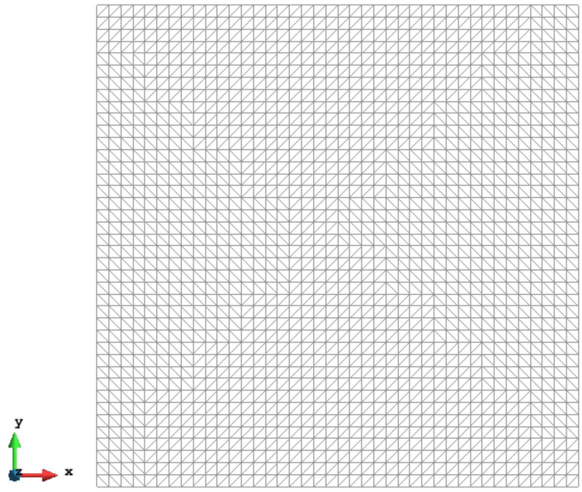


Figure 3 - Mesh structure (seen in the XY plane from the top)

Pressing on the "Create a MatLab file", it's been possible to generate a file that contains all the information of the model that need to be used in the MatLab code.

1.3 Working in the MatLab code

The MatLab file created by GiD presented one imprecision: the name of the matrix containing the Dirichlet BC for the code had different name that the MatLab code didn't recognize, so a change of name for this matrix was necessary. Furthermore, for the last part of the MatLab code there is a Function which creates an output file, that the GiD program reads, in order to show the post-process results. It turned out that the displacement vector *u* was shown as a "sparse double" entity, and the MatLab function "fprinf" used many times in the Function doesn't recognize it and gives error. It was necessary to convert this sparse double entity into a classical vector.

1.4 Post-process in GiD

Once created the ".flavia.msh and .flavia.res" files by the MatLab code both files have been opened inside the GiD problem. Now it was possible to show the results of the calculation (displacements and forces).

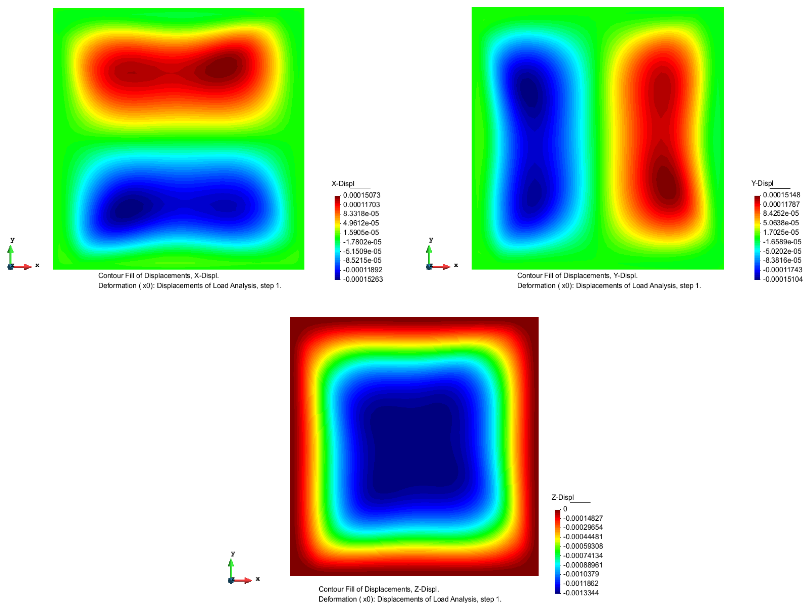


Figure 4 - Displacements from XY plane (top left - dispX / top right - dispY / bottom - dispZ)

As it can be seen in the figure 4, the displacement field (in all the directions) shows a symmetric distribution. While the z displacements have maximum value on the central core of the structure, as easily imagined and expected, the x and y displacements show their maximum values in the middle of each half of the structure (in the sense that displacements along x direction follow an halved structure along the x axis, while the displacements in y follow the same logic in the other direction). All this results follow the logic behind the own-weight loaded problem and the particular shape of the structure.

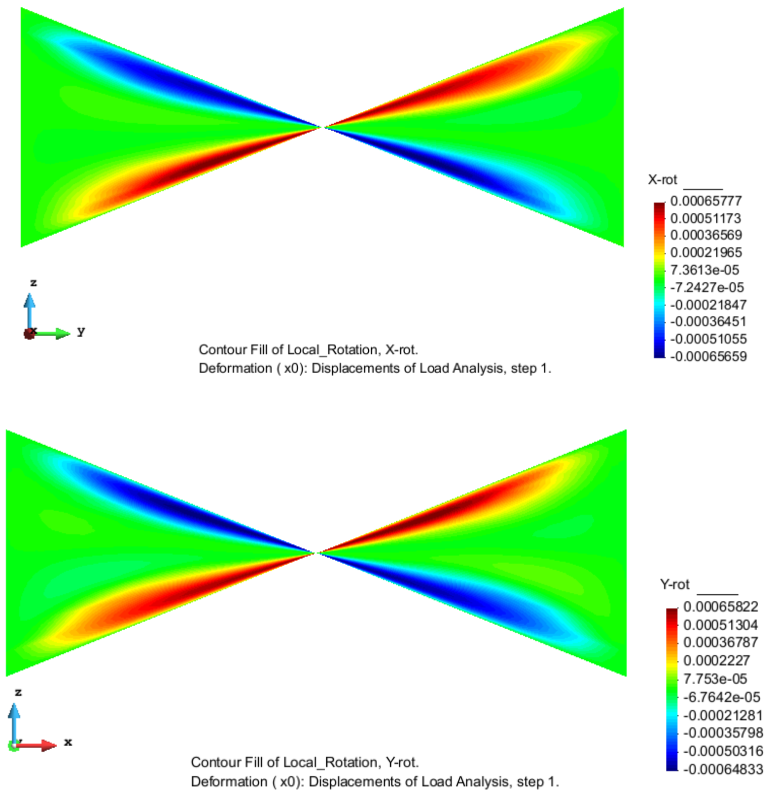


Figure 5 - Rotations (top - rotations around X seen from YZ plane / bottom - rotations Y seen from XZ plane)

The figure 5 shows the distribution of the rotations around X and Y axes. As expected and according to the shape of the structure, the load problem (self-weight) and the boundary conditions, both fields show a symmetric distribution and show the maximum values close to the edges (in the sense that the rotations around X are symmetric respect to the Y axis, while the rotations around Y are symmetric respect to the X axis). In figure 6 this logic is more underlined and visible.

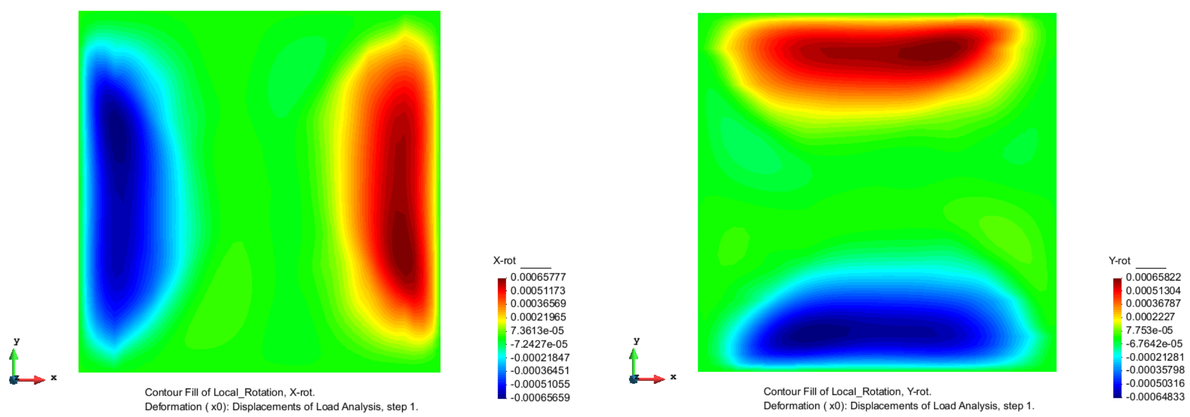


Figure 6 - Rotations seen from the XY plane (left - rotations around X / right - rotations around Y)

Concerning the forces, it's possible to see that the membrane problem is more significant than the bending and shear problems.

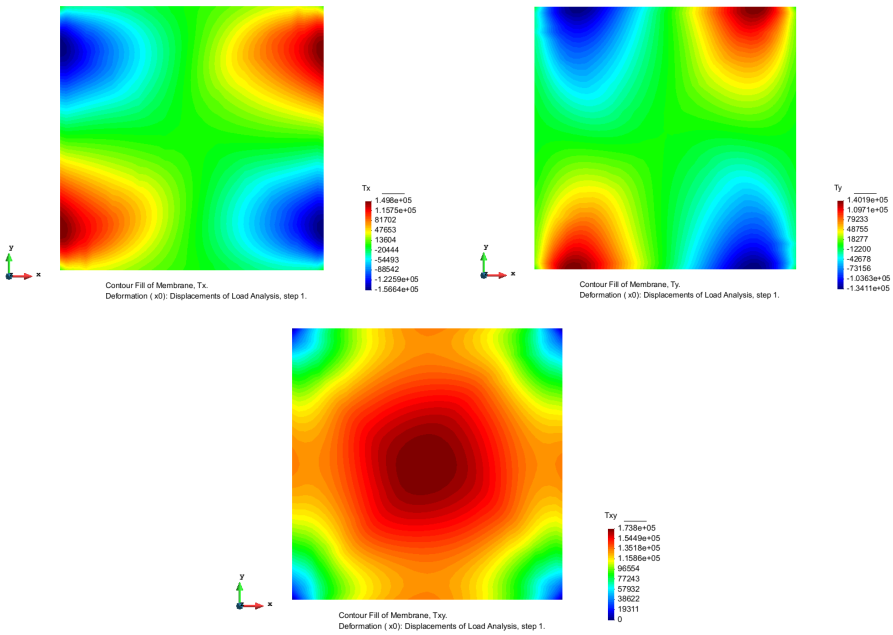


Figure 7 - Membrane forces seen from XY plane (top-left - Tx forces / top-right - Ty forces / bottom - Txy forces)

As shown in the figure 7, all three membrane forces (X, Y and XY) show a symmetric behavior. Due to the shape of the structure and the boundary conditions, the biggest values of Tx and Ty are close to the edges in positive and negative sign, changing very fast the value between these two extremes (for Tx = +1.498 e +05 / -1.566 e + 05 and Ty = +1.4019 e +05 and -1.3411 e +05). The Txy forces, instead, show a double symmetrical behavior (along both x and y axes), presenting the biggest value in the middle, where we have the area of change in shape and curvature of the structure and the value of 0 at the corners. This maximum value for Txy is 1.738 e +05.

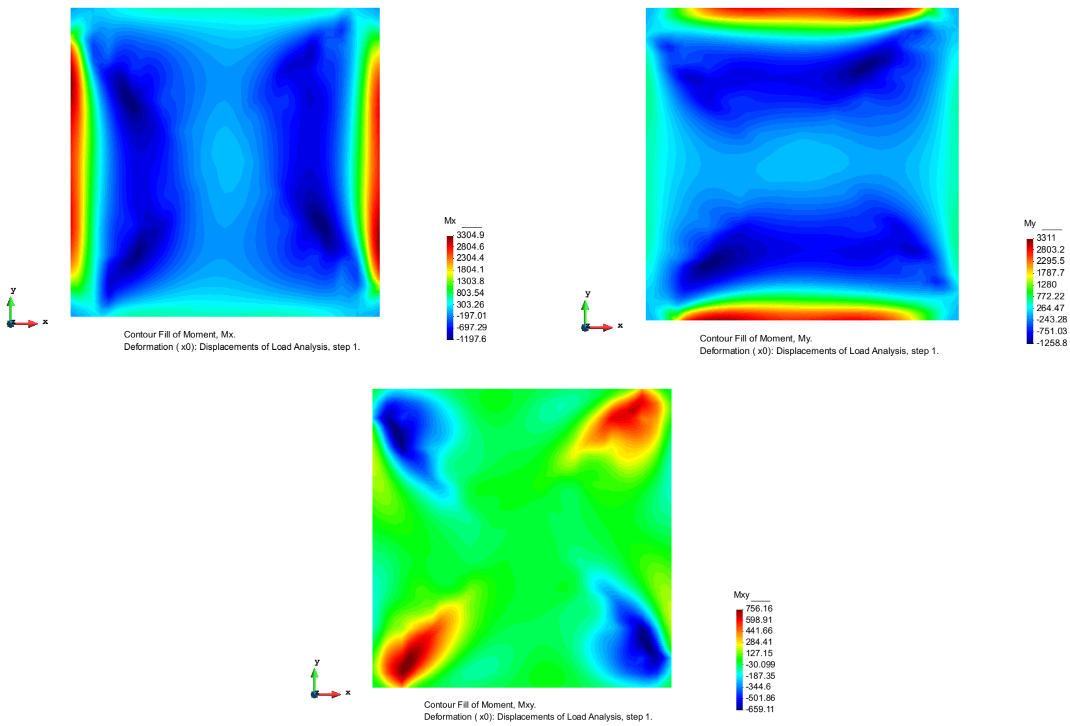


Figure 8 - Bending moments seen from the XY plane (top-left - Mx / top-right - My / bottom Mxy)

The bending moments show the same symmetrical logical behavior of the membrane forces, but present the biggest values of the moment exactly at the boundary edges of the structure (Mx = 3304.9 while My = 3311). *This is due to the fact that the structure is fixed in both directions so we can see it like a crossing beams frame. The not perfect symmetrical behavior is due to the shape of the structure that doesn't have the same geometric curvature everywhere. This quasi-symmetric behavior is confirmed in the torsional moment Mxy plot, that shows again two directions of symmetry, which are along the diagonals of the structure. The significant values of Mxy in the corners of the this model is due to the torsional effect that the gravity creates into this particularly shaped structure (Mxy = +756 / Mxy = -659).

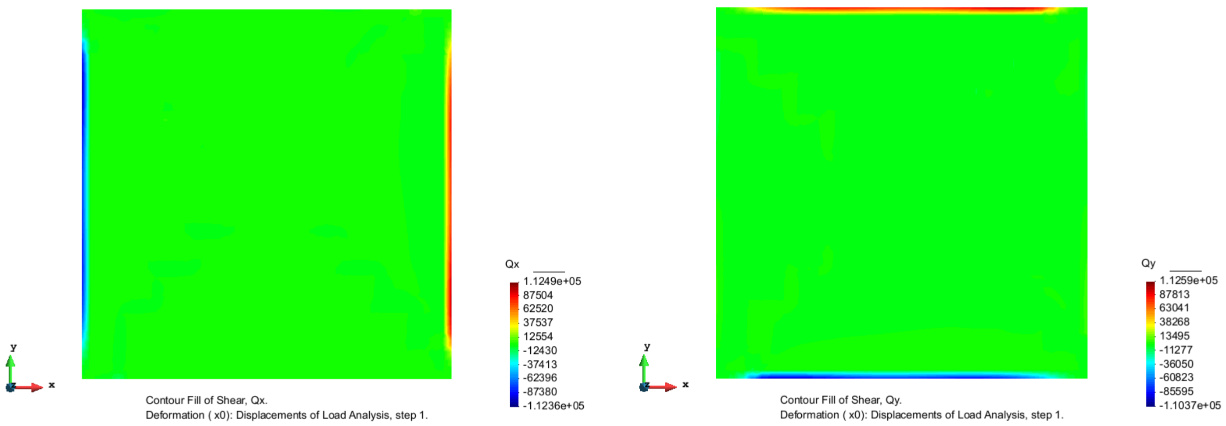


Figure 9 - Shear forces seen from the XY plane (left - Qx / right - Qy)

For the same reasons described for the bending moments distribution (*), it's possible to see in figure 9 that the shear forces show a symmetric behavior: as expected, it shows opposite big values at the edges of the structure following the same logic described before. (Qx = +1.1249 e +05 / Qx = -1.1236 e +05 and Qy = +1.1259 e+05 / Qy = -1.1037 e+05).