Student: Juan Diego Iberico Leonardo

Homework 8: Analysis of shells with plate elements.

Analyze the following concrete hyperbolic Shell under self-weight. Explain the behavior of all the Stresses presented. t = 0.1

Computational process:

- The first approach is creating the hyperbolic mesh in GID (software) giving as input a txt file with the nodal coordinates. In order to create the nodal coordinate matrix a linear interpolation is needed from the four giving nodes on the hyperbolic mesh of 3 node triangular elements.
  The boundary conditions applied to the structure are: clamped supports on the four edges. The load definition is only self-weight, and the material properties as concrete of thickness t=0.1.

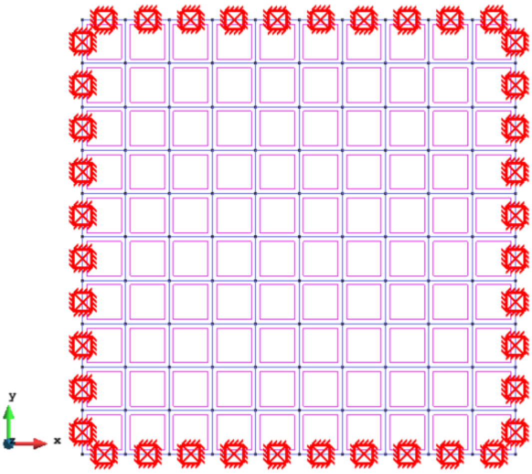

Figure 2: Boundary conditions (clamped edges)

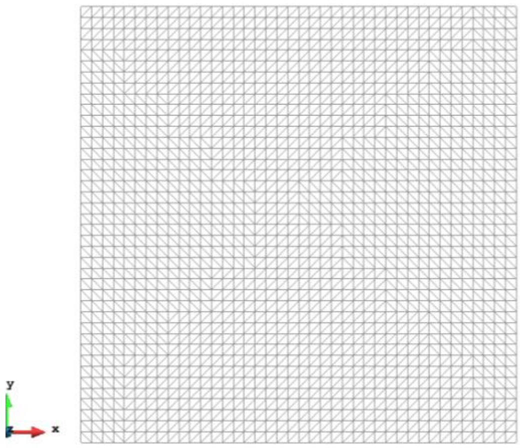

Figure 1: Mesh definition with 3 noded triangular

- The second step is based in compute the unknown vector (dispalcements and rotations) using the matlab code Lamina_T_RM to solve for 3 node laminar element Reissner-Mindlin.
- Once Matlab has solved the problem, using the function ToGid_Shell.m it is defined an input file in order to visualize the post-process in GID.
- The last approach: visualize and interpret the different results for the hyperbolic mesh:

*In order to ensure the correctness of the structure, it is needed to verify the surface normal components for each element and check that all the normal goes to the same direction (for the assembly in local coordinates).

Results analysis:

In order to ensure the correctness of the model, the graphic results have to should a coherence for all the stresses or displacements. The coherence on the results is obtained through the symmetry of the plots ensuring the to obtain the same local coordinates in all the elements (for the assembly process).

On the other hand, it is important to analyze the behavior of the structure/elements like membrane for example (when plane forcers are more important than bending moments).

Displacements:

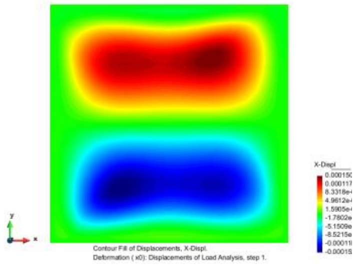
*Figure 3: **x** disp.*

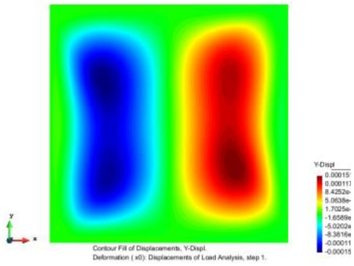
*Figure 4: **y** disp.*

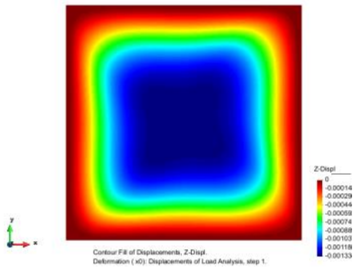
*Figure 5: **z** disp.*

As it can be seen it the figures presented, the coherence of the results are satisfactory, with the maximum value for w displacement at the center of the structure (under self-weight). On the other side, the displacement in each direction x and y are related with maximum positive and negative values depending on the curved face.

Bending Moment:

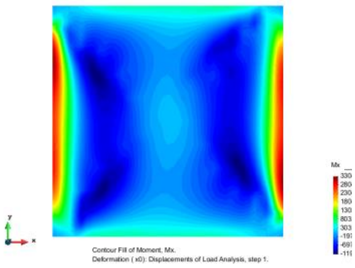
*Figure 6: **Mx***

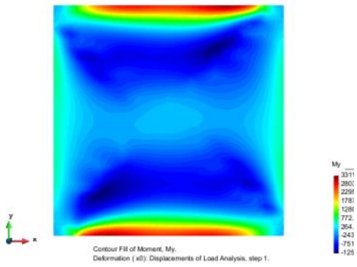
*Figure 7: **My***

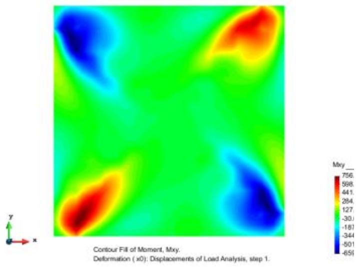
*Figure 8: **Mxy***

The results for the value of bending moment are not really significant for this concrete slab of 0.1. To sum up, the values obtained Mx = 3304, My = 3311 and Mxy = 756 doesn't represent the critical value for this kind of structure, leading to a membrane behavior.

In-plane stress:

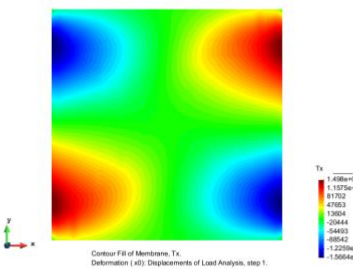
*Figure 9: **Tx***

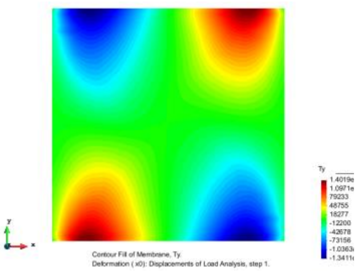
*Figure 10: **Ty***

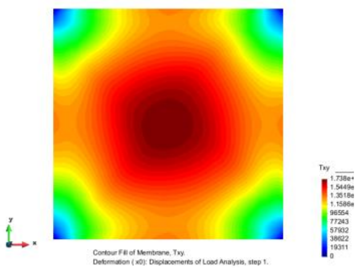
*Figure 11: **Txy***

In this last part of the analysis we can check the in-plane stress values for Tx, Ty and Txy. The values for these stresses are significant high compared with bending moment. The magnitude of this membrane results are Tx = 1.566×10^5, Ty = 1.4019×10^5 and Txy = 1.738×10^5.

To conclude, the model verification it is ensured, doing the K assembly in local coordinates and getting symmetric results. On the other side the structural behavior correspond to a membrane, with important values on plane-stress.