HomeWork - 1
Computational Structural Mechanics and Dynamics
Master of Science in Computational Mechanics 2016

Jorge Balsa   Paris Dilip Mulye

March 5, 2016

Exercise 1

Figure 1 and 2 summarize results with different mesh sizes and element types. The given values of σ_{yy} and u_y are used to calculate % errors in stress and displacement. In reality, relative errors can be used instead of absolute errors.

It is observed that displacement convergence is achieved faster than stress convergence, which is expected since σ contains the derivatives of u. Figure 2 indicates that if sufficiently more nodes are used, difference between lower order and higher order element is less. From Figure 1, it can be concluded that at lower numbers, a Tri with 3 nodes performs poorly even compared to a Quad with 4 nodes.

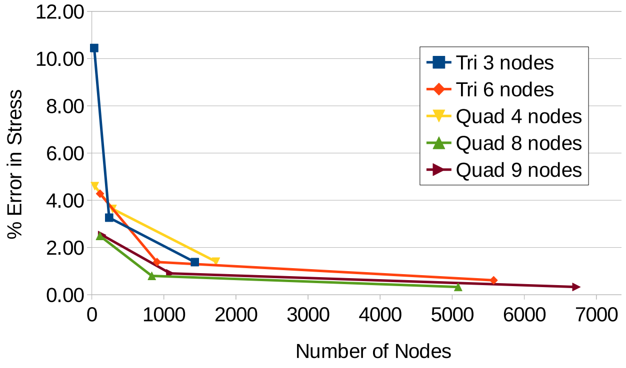

Figure 1: σ_{yy} Convergence

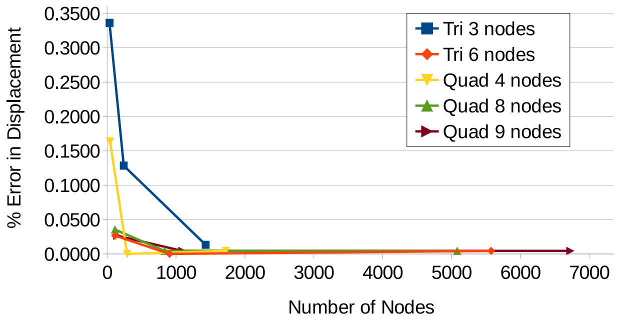

Figure 2: u_y Convergence

Figure 3 and Figure 4 depict the u_y and σ_{yy} when meshed with Quad with 9 nodes along with mesh plot. Three meshes were generated for every element type using mesh sizes 0.75, 0.25 and 0.1. Since thickness of the plate is very less compared to other dimensions, plane stress assumption used in simulation is justified.

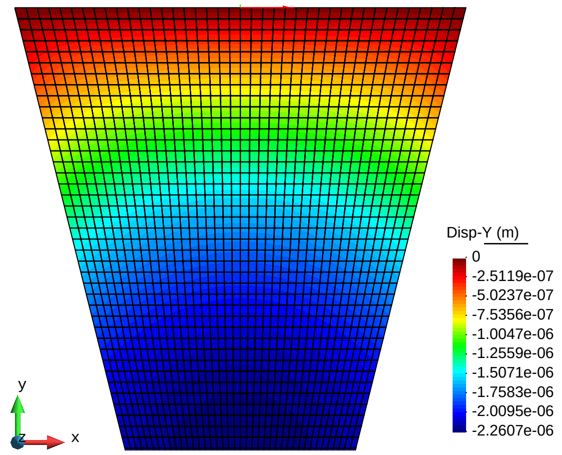

Figure 3: Displacement Plot: u_y

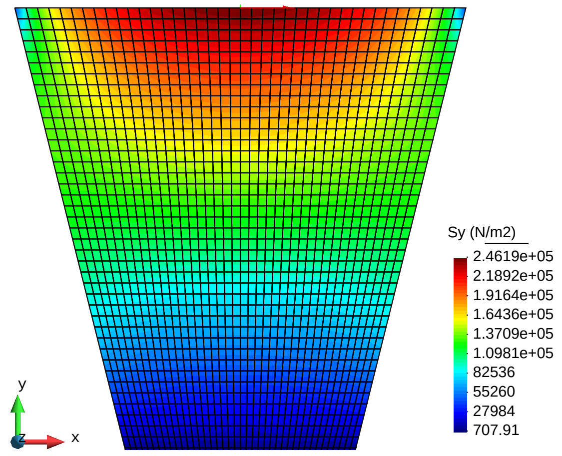

Figure 4: Stress Plot: σ_{yy}

Exercise 2

The case without sag in foundation was simulated and compared to the one with sag. The interpretation of physics of problem as per authors is as follows; when the foundation is strong, all three columns will share load. This is simulated using a fixity in both X and Y directions for all three columns. When the foundation undergoes a sag below central column, the central column will no longer be able to take part in load sharing since it has no strong base to provide reaction. This case is simulated by keeping the bottom edge of central column to be free in both X and Y.

Both models were simulated using same mesh and mesh size of 0.1 was used to create Tri 3 elements with total 4227 nodes. Figure 5 and 6 show displacements in both cases. Figure 7 and 8 show von Mises stress distribution for both cases.

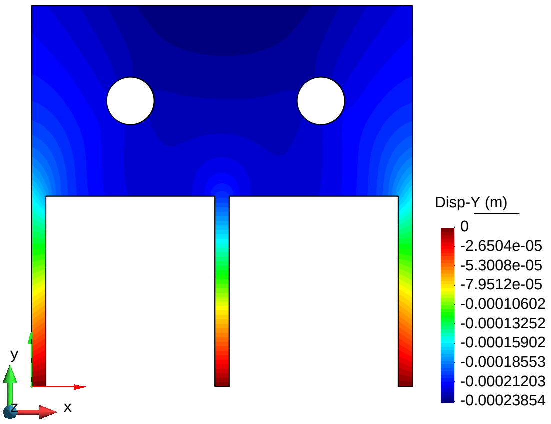

Figure 5: Displacement u_y (Without Foundation Sag)

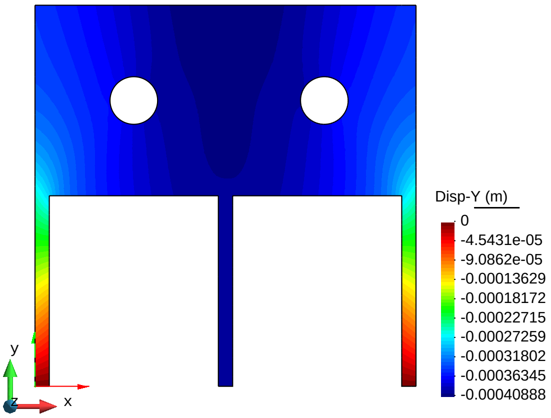

Figure 6: Displacement u_y (With Foundation Sag)

The displacement of top part of structure in case with foundation sag is more by 0.17 mm. This behavior is expected, since when foundation sags, there is less support for structure and its stiffness is less. Therefore, it is expected to deform more.

The maximum von Mises stress was found to increase by 62.4% in case of foundation sag. Also stress distribution is found to change significantly. This is justified with the fact that initially, majority of load was going through the central member (shortest loadpath) and it had the highest stress. However, due to foundation sag, this central member can no longer support any load and loadpath should be diverted to other two supports which now have to support more load than before. Therefore, the highest stress now occurs on them.

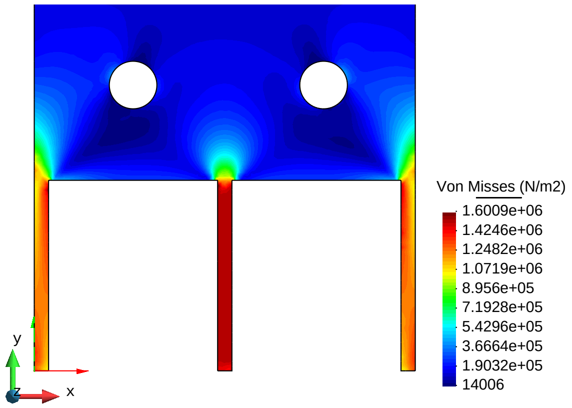

Figure 7: von Mises Stress (Without Foundation Sag)

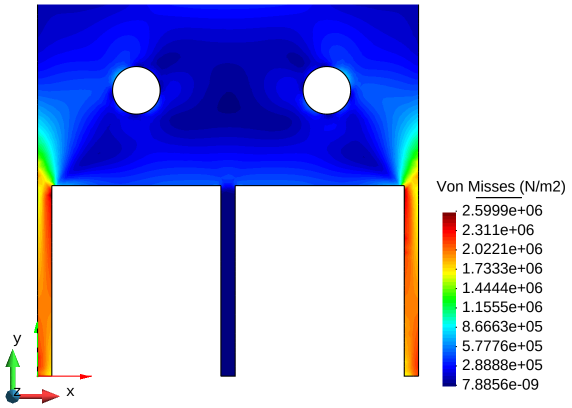

Figure 8: von Mises Stress (With Foundation Sag)

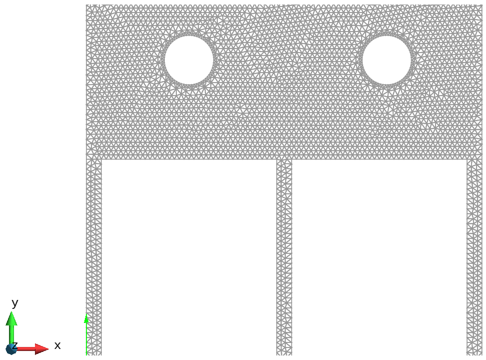

Figure 9: Mesh Plot

Exercise 3

The modeling of the geometry and mesh holds quite some importance for this exercise. For ease of creation of steel reinforced sheets, layer functionality of GiD was used. Surfaces were split further to form rectangles in order to simplify structural mapping of mesh. Lines were created at critical locations (shown in Figure 10) so that supports (highlighted in red) can be simulated. Creating a point at a distance of 0.2m from either sides of the geometry ensured that a node is present at that location thus making the application of BCs easier.

The geometry was meshed with Quad 4 node element as shown in Figure 11. Nodes were merged so that load transfer can occur via both concrete and steel. This resulted in formation of multiple elements which have exactly same nodes. Materials were assigned to elements.

This being a 2D problem, all surfaces are modeled at the mid surface and all the mid surfaces overlap one another. Thus, in this case, authors do not see any difference between modeling 2 steel plates of thickness 8mm and modeling a single steel plate of thickness 16mm.

For this exercise, all the loads and boundary conditions were applied at FE level. Nodes corresponding to the red lines in Figure 10 were fixed in both X and Y directions. Total Force was calculated using given linear force density and integrating it over the length of application of force. This total force was then divided by the number of nodes and obtained value was applied at every node.

Note that a better simulation would be to apply this as a pressure on elemental faces. In reality, as elements deform due to pressure, the direction of pressure changes in order to maintain normal to face property of pressure. This is not the case with force which would always maintain its direction. But since, this is a small strain and linear problem, both pressure and force would create same result.

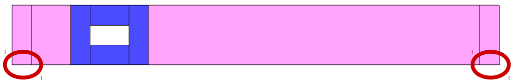

Figure 10: Geometry and Support Locations

Figure 12 and 13 show the plots of von Mises stress. The displacement in y direction is shown in Figure 14. The maximum von Mises stress was found to be 2.375E7 Pa. The maximum displacement in -y direction was found to be 0.534 mm.

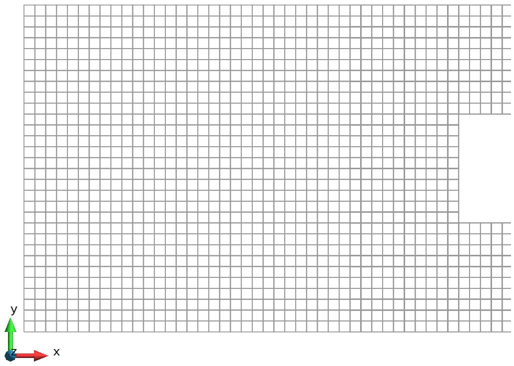

Figure 11: Mesh Plot

Figure 12: von Mises Stress Plot

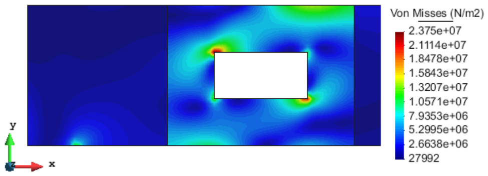

Figure 13: von Mises Stress Plot: Zoomed

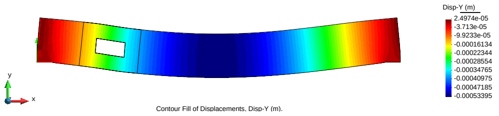

Figure 14: Displacement Plot: u_y (Scale 400X)

Exercise 4

It is assumed that width of the tank is very big (in Z direction) and thus there would not be any considerable strain in Z direction. This justifies plane strain assumption. Also position of left end of cross section has not been given. It is assumed to be a big number comparatively (5m). The left end of the cross section was fixed in X direction (due to symmetry) and kept free in Y.

Figure 15 shows application of loads and BCs. The ground is simulated using the given elastic property. The lines highlighted in red were given stiffness in direction Y. The line highlighted in blue was given appropriate components of stiffness so that resultant stiffness in direction of normal to edge matches given value.

The force of static water is only the pressure. Pressure on bottom of tank was calculated using $P = \rho g H$ and thus a constant pressure load was applied on orange line. Pressure on the side face of the tank (indicated by green line), varies linearly with depth. This was applied at FE level.

The geometry was meshed using Quad with 4 nodes. Figure 16 shows the mesh plot. Therefore, for every element on the green line, a pressure was calculated and applied on individual element faces.

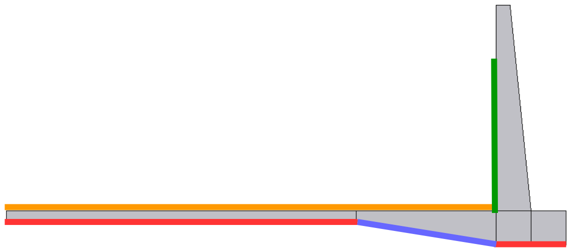

Figure 15: Loads and BC

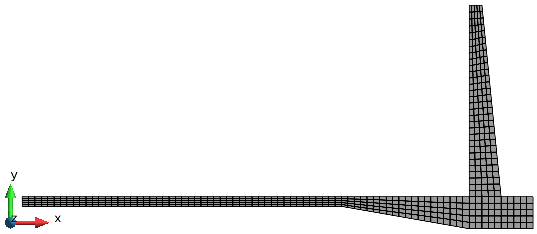

Figure 16: Mesh Plot

Figure 17 shows von Mises stress plot. The maximum von Mises stress was found to be 5.6095E6 Pa. Figure 18 shows Y displacement plot. The maximum downward displacement was found to be 3.618 mm and maximum upward displacement was found to be 0.495 mm.

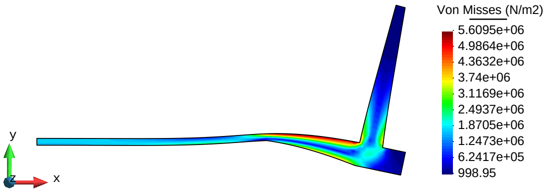

Figure 17: von Mises Stress

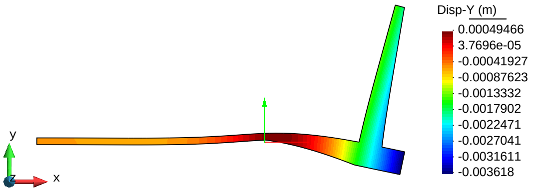

Figure 18: Displacement Plot

THE END