SIGMA/W Tutorial

Introduction

This is a fairly simple introductory example, so that it can be analyzed with the GeoStudio Student Edition. The primary purpose is to demonstrate to new SIGMA/W users how to get started, to introduce the typical data required to do a SIGMA/W analysis, and to illustrate what can be done with SIGMA/W. This problem focuses on computing stress changes in the ground beneath a cylindrical, fluid-filled tank.

Numerical Simulation

When studying a problem that is symmetric about a vertical axis, the three-dimensional effects can be analyzed with a two-dimensional finite element mesh, as illustrated by the following sketch. A special axisymmetric formulation in SIGMA/W for this case considers the stresses in the X-Y plane, as well as the circumferential stresses.
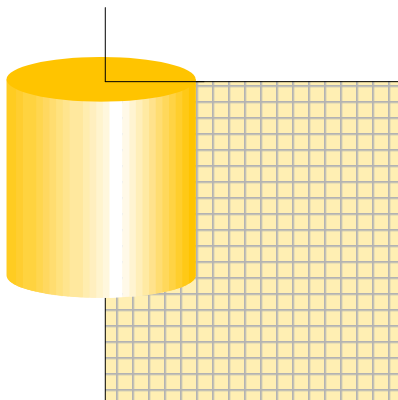

Figure 1. Visualization of axisymmetric analysis in 2D.

Figure 2 shows the corresponding SIGMA/W problem setup. The diameter of the tank is 5 m and the applied pressure is 40 kPa. The soil region is 40 m wide and 25 m high. The upper 5 m of soil has different properties than the underlying 20 m. The weight of the tank must be applied as boundary condition and specified boundary pressures must be applied on region edges. Consequently, a Point is added at the location of the outer edge of the tank (Point 10).
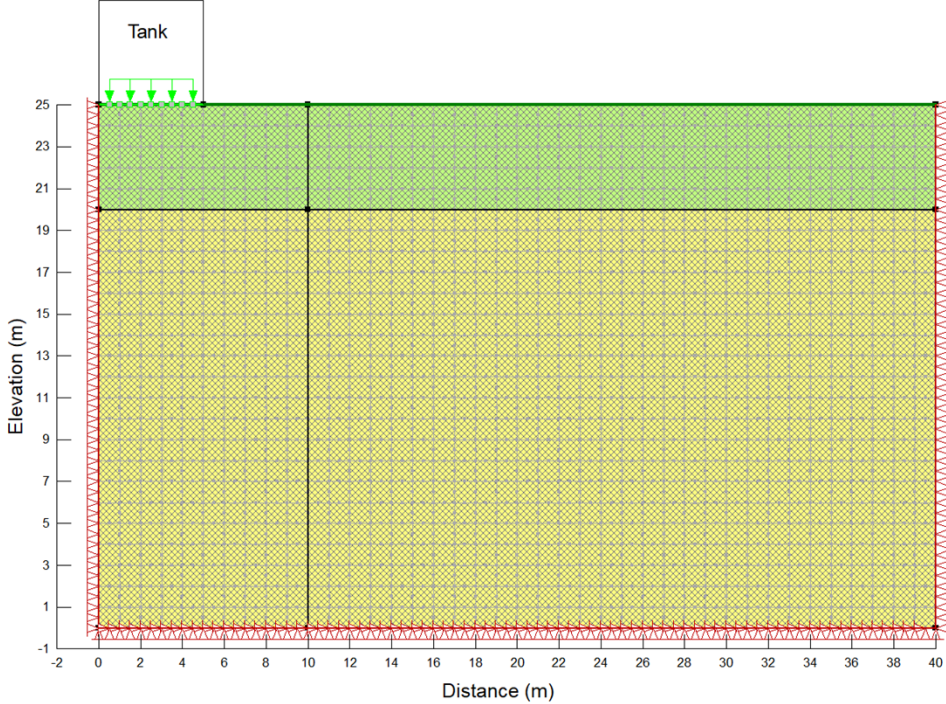

Figure 2. Problem configuration.

Along the left symmetric axis, we want to allow vertical movement (settlement) but no lateral (horizontal) movement. Along the bottom and right vertical side, it is considered that these boundaries are far enough away from the applied surface pressure that they can be fixed in both the X and Y directions. SIGMA/W has default boundary conditions you can use for this (Fixed X and Fixed X/Y, respectively).

The tank pressure is specified as a Y-stress boundary with a magnitude (action) of -40 kPa. The negative value means it is pointing downward. Now it is evident why it was necessary to subdivide the region edge so that the tank pressure is applied only under the tank.

The soil will be treated as being isotropic elastic with the following properties:

Upper soil: E = 30,000 kPa and Poisson's ratio v = 0.334

Lower soil: E = 50,000 kPa and Poisson's ratio v = 0.334

The response type is defined as undrained, which will simulate a perfectly undrained pore-water pressure response to the loading.

Now that the geometry, boundary conditions, and material properties are specified and applied, we should look at the mesh and make modifications if necessary. This is done using the Draw Mesh Properties command. A global element size of 1 m is used in this analysis. A decision was made to use 8-noded higher order elements in all of the regions. The resulting mesh is shown in Figure 2.

Results and Discussion

The next step is to click on the SOLVE button and compute the solution to the finite element equations.

Vertical stresses

Earlier it was noted that the object here is to compute the stress changes in the ground due to the weight of the fluid in the tank. The best way to look at this is to create a contour plot of the Y-stresses, as in Figure 3. The contour labels are in units of kPa. The contours from a pressure bulb, such as are typically shown in most introductory Soil Mechanics textbooks, suggest that the analysis has been correctly set up.

Note that the 40 kPa contour right under the footing is somewhat irregular and squiggly. This is primarily due to numerical noise and nothing to worry about. Ideally, the 40 kPa contour is right at the contact, but in a numerical approximation such as a finite element analysis it is not possible to achieve this ideal condition.
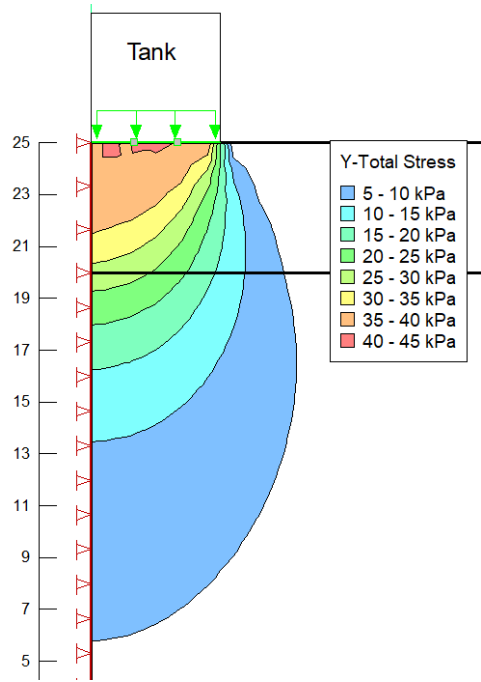
Figure 3 . Vertical stress change contours.

Vertical settlement

Figure 4 shows the vertical settlements as a deformed mesh at a 300X exaggeration. The maximum settlement is about 0.006 m (6 mm) under the center of the tank.
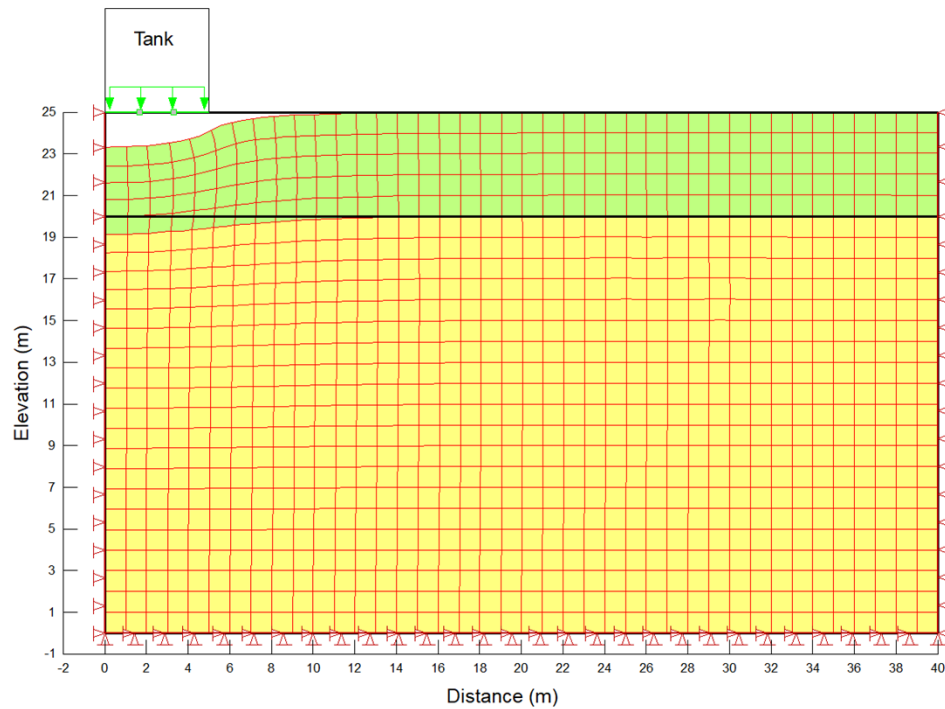

Figure 4. Settlement as a deformed mesh at 300X exaggeration.

Graphing

In addition to looking at the results as contour plots or deformed mesh plots as above, there are multiple ways of looking at the results with the Draw Graph command in SIGMA/W. Figure 5 shows the computed settlement along the ground surface. Figure 6 gives the lateral (x) displacement along a vertical profile at the outer edge of the tank.
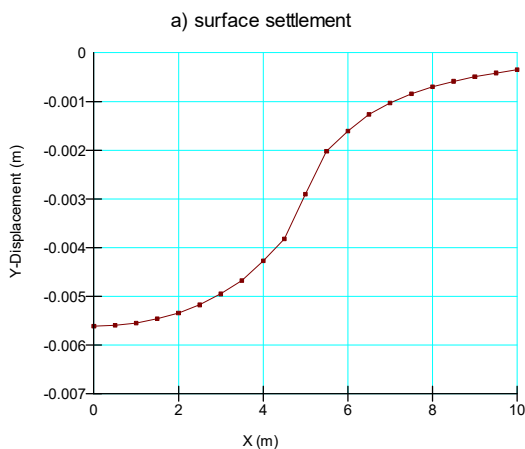

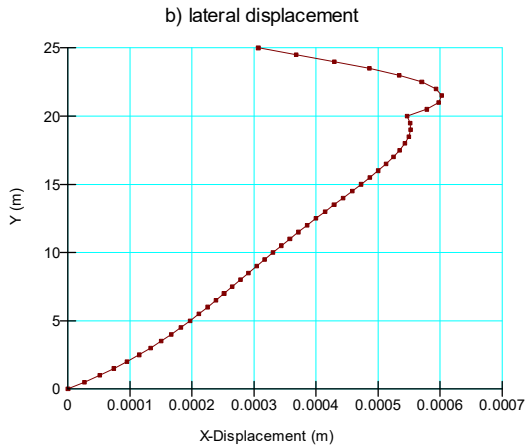**Figure 5. Ground surface settlement profile.**

b) lateral displacement

Figure 6. Lateral displacement profile.

Summary and Conclusions

This simple introductory example has all the components of a finite element analysis, which are primarily geometry, boundary conditions, and material properties. The example also illustrates how this type of problem can be easily and conveniently analyzed with SIGMA/W. Moreover, it clearly demonstrates how the results can be visualized with colorful contour plots and graphs.

If you take the time to duplicate this analysis by going through all the steps, you will have a very good concept of what is required to do a SIGMA/W analysis, and what can be achieved with this powerful software product.