Structural Analysis and Design of Tank Structures in STAAD.Pro V8i

By

Bentley Structural Group

September 25, 2009

STAAD.Pro V8i can be used to analyze and design water tanks. Figure 1 shows a steel tank that has been modeled using STAAD.Pro V8i. The geometry of this tank was drawn using the STAAD interface. The commands such as ***perform circular repeat***, ***create in-fill plates*** were very helpful in generating the geometry.

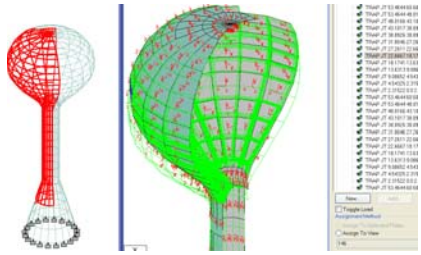


Figure 1: Steel Tank geometry in STAAD.Pro

Figure 1 also shows the local co-ordinate axis of all the plates in the graphics. In order to define the direction of the water pressure loading on the plates, the user must use the plate local co-ordinate axis as the reference. In STAAD.Pro, pressure loads on plate elements can be applied with respect to the global and local co-ordinate axis. The orientation of the plate local co-ordinate axis (i.e. z-axis) for all plates must be pointing towards the center of the tank or away from the center of the tank. When the in-fill plates command is used, it is very likely that the z-axis for some plates is pointing towards the center and some are pointing away from the center. This problem can be easily rectified using the reference point command in STAAD.Pro.

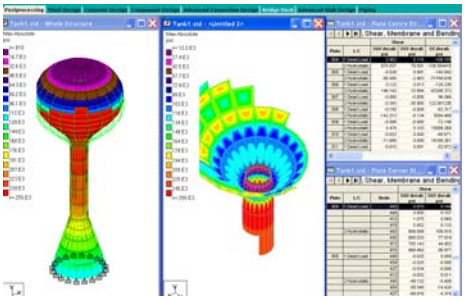


Figure 2: Stress distribution diagram

Once all the loads have been applied, the engineer may create ***groups*** of plates and assign variable steel plate thickness to them. These groups are very helpful when looking at the stress tables in STAAD.Pro ***post-processing mode***. The post-processing mode in STAAD.Pro allows users to look at steel stress distribution diagrams and deflection diagrams. The deflection diagrams can be exaggerated to verify if there are any connectivity/instability issues in the model. The stress distribution diagrams shown in Figure 2 are displayed with a color legend and the finite element analysis (FEA) results. The engineer can simply click on a high stress zone in the graphics and STAAD.Pro will pin-point the FEA results for the corresponding plate in the results table.

The results in the FEA results tables can be filtered. For example, one may only be interested in the results for the combination load cases for plates connected to the foundation. Using a combination of the filter tools and plate grouping options discussed above, one could easily obtain these results and compare them with the allowable stress values.

STAAD.pro also provides the user with dynamic analysis capabilities. An engineer may easily find out the various modes and associated frequencies of a steel tank using the ***MODAL CALCULATION REQUESTED*** command in STAAD.Pro. The modes shapes will be plotted in the STAAD.Pro user interface as illustrated in Figure 3. An engineer has the option of doing a code based seismic analysis or a full seismic ***time history analysis*** on tank structures. The seismic time history analysis could be based on ground motion data (i.e. time vs. ground acceleration) from a past earthquake and STAAD.Pro will be able to calculate the maximum base shears for the support nodes. The time vs. displacement, acceleration, and velocity graphs will be available to the user for each and every node in the model.

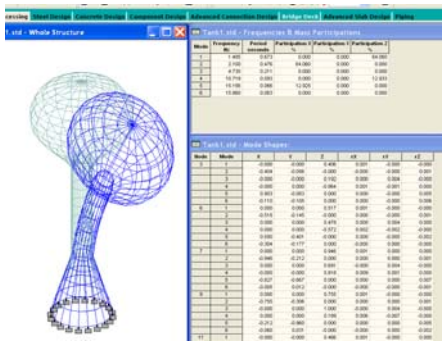


Figure 3: Results of dynamic analysis

In addition to steel plate tanks discussed so far, engineers may also easily model concrete tanks

using STAAD.Pro as shown in Figure 4. The hydrostatic load generator in STAAD.Pro was used to load the vertical concrete plate elements. STAAD.Pro's concrete element design tool was used to calculate the area of steel required for flexure as per the ACI 2005 code.

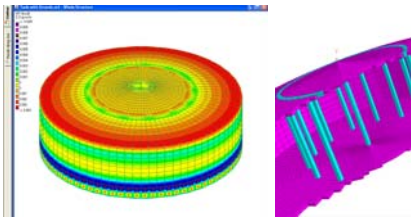
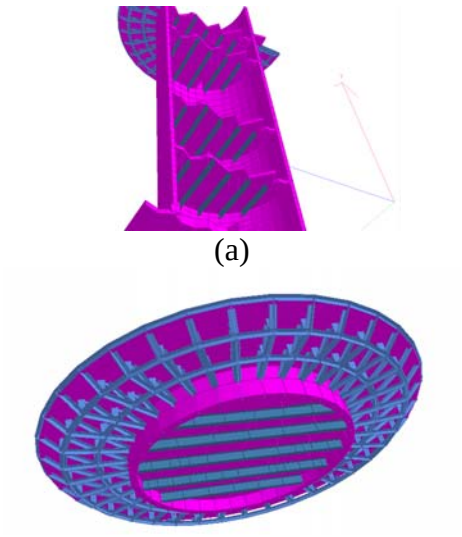


Figure 4: Circular concrete tank

Occasionally a water tank as shown in Figure 5(a) may also have steel beams attached to it as illustrated in Figure 5(b). These steel beams can be designed as per the AISC 13th edition code in STAAD.Pro.

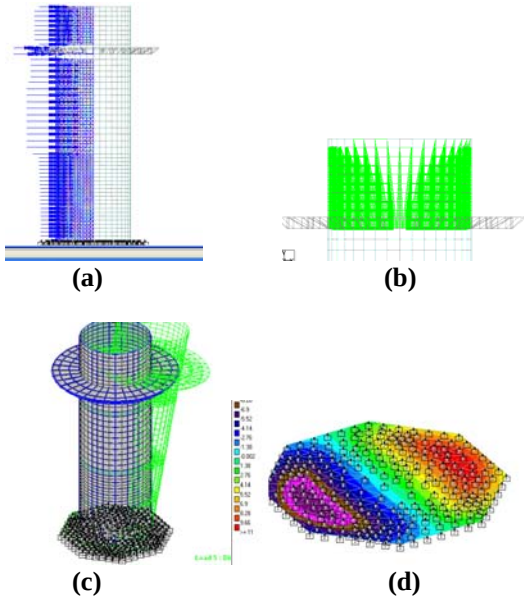


(a)

(b)

Figure 5: Concrete Deck supported by Steel Sections

The tank model shown in Figure 5 was constructed on top of a MAT (Raft, slab-on-grade) foundation. This MAT foundation was modeled using plate elements in STAAD.Pro. STAAD.Pro has a spring support generator for MAT foundations modeled as plates. This support generator basically generates spring constants for each and every node on the plate MAT depending upon the sub-grade modulus of the soil and the influence area of a node point. The user may also opt to specify the springs as "compression-only" springs and see if the footing will experience any uplift in a seismic or high wind load event. Figure 6 (a) illustrates the wind load generation on a concrete tank as per the ASCE-7-2002 guide. Figure 6 (b) illustrates hydrostatic load generation on the vertical plate elements.



(a) **(b)**

(c) **(d)**

Figure 6: Circular concrete tank

Figure 6 (c) illustrates seismic load response on of the structure and Figure 6 (d) shows the resulting base pressure. The engineer may compare these base pressures with the soil bearing capacity.

The MAT foundation shown in Figure 6 (d) can be exported out to STAAD.foundation. The engineer will be able to see the MAT rebar layout in the longitudinal and transverse directions for both top and bottom layers in STAAD.foundation.

STAAD.Pro's tank modeling features discussed above have been used to analyze and design water tanks, tanks used in the water purification and sewage industry, tanks in the nuclear energy production and nuclear waste storage industry.