Staged Construction of an API 650 Tank on a Settling Foundation

Alexander Hay, Rudy du Preez, Roelof Minnaar
Advanced Structural Mechanics (Pty) Ltd, CSIR Campus, Meiring Naude Road, Pretoria, South Africa

Abstract

The staged construction of an API 650 tank is analyzed in this paper. The tank is considered to be erected on a foundation that is initially irregular, and which then experiences differential settlement during the course of construction. The consequence of foundation settlement and initial irregularity is that the final shape of the tank deviates from the intended cylindrical shape.

During construction, each strake is added to the tank by positioning the (relatively flexible) rolled plate segments on the top edge of the strake below, while at the same time maintaining a fixed gap between vertical edges of adjacent plates. Once positioned, the vertical welds between plates are executed, and then the circumferential weld to the strake below. As such, the strakes can be considered to be placed on top of each other in a deformed (as dictated by the shape of the top edge of the strake below), but essentially stress-free manner. The structural response of the tank as a whole is also affected by a wind girder, which is moved up the tank in a sequence of temporary positions as the strakes are added, until it is fixed in its final position to the top strake of the tank. In its sequence of temporary locations, the wind girder not only provides stability to the tank shell, but is also used to assist in maintaining the cylindrical shape of the tank.

Analysis was made possible by LS-DYNA®'s staged construction functionality. In addition to the standard staged construction keywords, special provisions were required to enable sequential placement of the strakes in deformed but stress- and strain-free states. This was accomplished by running a sub-problem for each successive strake, where the displacement boundary conditions of the sub-problem enforced shell displacement continuity between the current strake and previous strake. Once the constructed shape of the current strake had been determined, it was transferred to the main model. The strake part was then activated in the main model for further analysis, as further foundation differential settlement occurred. Data transfer between the main model and sub-models was accomplished by means of custom Python scripts.

The analyses allowed for the quantification of the effect of differential foundation settlement and initial foundation irregularity (while also including the effect of the wind girder) on the final constructed shape of the tank.

Introduction

LS-DYNA is used to analyze the bottom-up staged construction of an API 650 [1] storage tank. The tank is constructed on a foundation which experiences differential settlement during the course of construction. In addition to this, the foundation is not level at the start of construction.

The aim of the analyses is to determine the relative effects of the initial foundation irregularity and foundation settlement during construction on the final shape of the storage tank.

Due to the physical method of tank construction, a novel approach to the analyses is required, wherein a sequence of sub-analyses are run and used together with the LS-DYNA staged construction functionality to model the tank as constructed.

Tank geometry and construction methodology

As shown in Fig. 1, the tank is 36m in diameter and 24m high, and is constructed in 10 strakes, with a wall thickness varying from 21.5mm at the base to 8mm at the top. A wind girder, made up of 12 of segments, is permanently installed on the top strake of the tank after installation of strake 10 (indicated in red). The same wind girder is attached in a sequence of temporary positions as the tank is erected (outlines indicated).

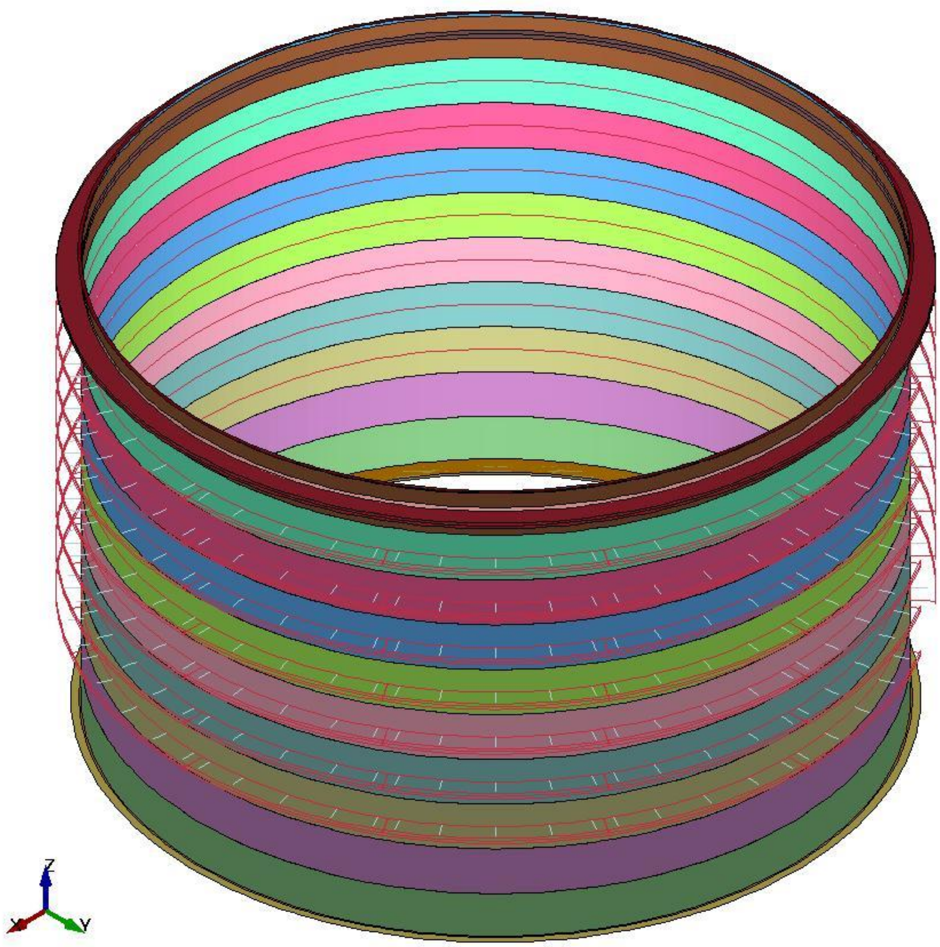

Figure 1: API 650 storage tank

During construction, each new strake is added to the tank by positioning rolled plate segments (11 segments around the circumference) on the top edge of the previous strake. The vertical edges of these segments are aligned (by moving the top of the segment radially inwards or outwards) so that there is a constant gap between the vertical edges of adjacent rolled segments. Once the segments have been set up, the vertical welds on that strake are performed, and finally the circumferential weld to the strake below. As such, the shape of each new strake is determined by the shape of the top edge of the previous strake, and any radial or vertical deviations from the ideal cylindrical tank shape tend to propagate to the new strake. Due to the strake wall thicknesses being thin in relation to the tank diameter, each strake is relatively flexible, following completion of the vertical welds and prior to being welded to the strake below. Due to the weld sequence, the strakes are assumed to be added in a deformed, but essentially stress-free state. For the ideal case where a strake is placed on a flat surface, the resultant geometry would be perfectly cylindrical.

From strake 3 upwards, the wind girder is temporarily installed on each strake as it is completed, to assist in maintaining the tank's desired cylindrical shape, and to also provide access for work on the next strake, and increase stability of the tank shell. When installing the wind girder, the tank shell is forced to follow the shape of the segments (within certain radial displacement limits, and at five points along each wind girder segment). This is practically achieved by forcing carrot drifts through lugs on the tank shell and wind girder. When installed in a temporary location, one segment of the wind girder is not installed, resulting in a gap in the wind girder (as seen in Fig. 2). The joints between the wind girder segments are modelled as hinges.

Figure 2: Construction up to third strake and temporary wind girder installation

The sequence of construction steps is illustrated for strakes i and i+1, and is summarized as follows

(a) Install wind girder temporarily at mid-point of strake i.

(b) Construct strake i+1 by aligning rolled segments on top edge of strake i, and ensuring adjacent vertical edges have a constant root gap. Perform vertical, and then circumferential welds.
(c) Remove wind girder from strake i.
(d) i=i+1, return to (a)

The above sequence is repeated, until strake 10 is completed, and the full wind girder is installed in its final position on that strake.

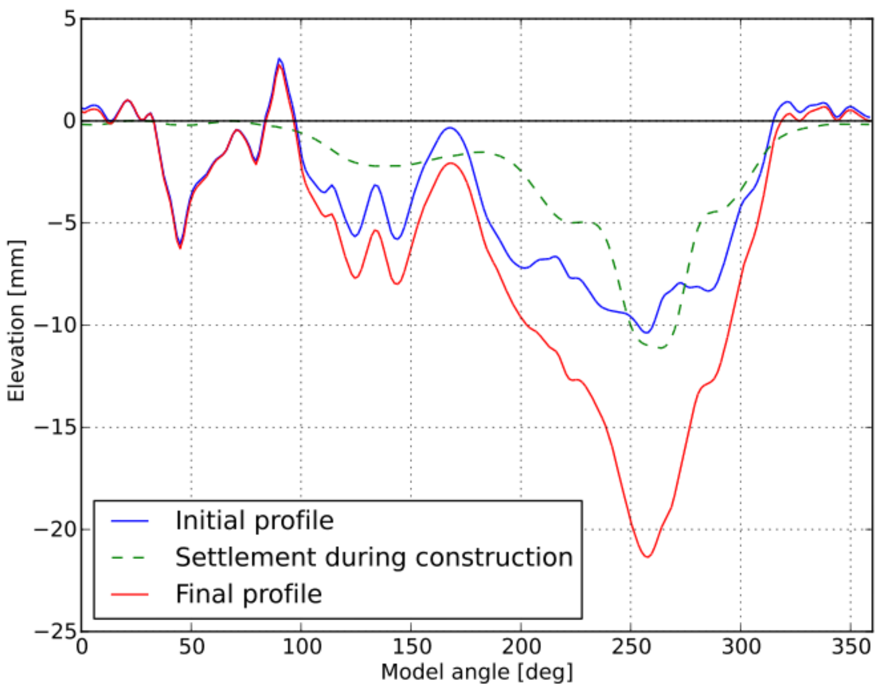

Figure 3: Analysis A – circumferential foundation profiles

The tank is built on a reinforced concrete ring foundation, which experiences differential settlement during tank construction, and which is also initially not level. In order to understand the separate effects of initial foundation imperfections and settlement during construction, two analyses were run. In the first analysis (analysis A), initial foundation irregularities were applied and in the second (analysis B), the foundation was assumed to be perfectly level. In all other respects the two analyses were identical. Fig. 3 shows the initial foundation profile, differential settlement profile, and consequent final foundation profile for analysis A. For analysis B only the settlement profile is applied, and thus the final profile for that analysis is the same as the settlement profile.

Tank finite element model and analysis approach

The finite element model of the tank is as shown in Figs 1 and 2. The wind girder is included in the model in all of its temporary positions as well as in its final position (as illustrated in Fig. 1). The wind girders are activated and deactivated sequentially from bottom to top as the staged construction progresses. For each wind girder location, it is necessary to model both the contact between the wind girder and the tank and the effect of pulling the tank towards the wind girder using the carrot drifts.

The continuous contact between the wind girders and the strakes was modeled using *CONTACT_NODES_TO_ SURFACE_INTERFERENCE. The effect of forcing the shell to comply with the wind girder shape (using the carrot drifts) was also modeled by means of a node to surface contact definition at the carrot drift locations. In this case the node set involved in the contact definition was a set of nodes located to the inside of the tank and connected by beam elements to the wind girder (beam elements are shown in light blue in Fig. 1). By varying the internal offset of these nodes from the tank shell, the clearances in the assembly can be simulated. This contact definition acts to limit the gap between the strake and wind girder, and is effectively inactive as long as the strake remains in close proximity to the wind girder.

The concrete foundation is itself not modeled, but the top surface of the concrete is represented using shell elements. The vertical displacements of the nodes are prescribed to follow the settlement profile in time. Contact between the top of concrete and tank floor allows the floor to lift off the concrete as required by the structural response.

Use of the LS-DYNA staged construction functionality allows for the sequential activation of each strake and activation and deactivation of the wind girder in various temporary locations, as construction progresses. Due to the construction methodology employed, the shape of each new strake is a function of the shape of the previous strake and can only be determined once the shape of the previous strake is known.

By means of the *CONTROL_STAGED_CONSTRUCTION keyword, the analysis can be stopped and restarted at the end of any defined construction stage (*DEFINE_CONSTRUCTION_STAGES), where the various model parts are defined as being active or inactive for various construction stages (*DEFINE_CONSTRUCTION_PART). Importantly, the solver writes out a keyword file named “end_stage???_dynain” at the end of each construction stage, which contains amongst other data, the nodal coordinates and stress and strain state of the elements at that time step. This provides a means for interrupting the analysis to calculate the shape of each new strake by means of a sub-model, and then restarting the analysis with the new strake geometry.

The geometry contained in the ten sub-models (one for each strake) is simply a cylinder of shell elements, with properties, nodal coordinates, node numbers and element numbers corresponding to that same strake in the main model. Displacement boundary conditions are applied at the bottom edge of the strake under consideration in order to simulate the strake being assembled on the top edge of the previous strake. Analysis of the sub-model yields the deformed/assembled shape of the strake, which is transferred stress-free to the main staged construction model, by updating the nodal coordinates of the current strake in the “end_stage???_dynain” file. As the staged construction analysis proceeds, the current strake is activated in its assembled shape, which effectively simulates completion of the circumferential weld to the strake below.

Python scripts were developed to ease data interfacing between the main staged construction model and the submodels. The first script reads the calculated displacement components at the top of the previous strake and writes out the *BOUNDARY_PRESCRIBED_MOTION keywords which are applied to the base of the current strake sub-model. A second python script reads the deformed nodal coordinates of the current strake at the end of the sub-model analysis, and updates the nodal coordinates in the main file to match these values. A third python script was developed to determine the resultant displacements of all the staged construction "end_stage???_dynain" files to allow for plotting and interpretation of results (d3plot displacements are from the start of current analysis – which is generally not the start of tank construction).

The staged construction analysis of the tank thus entails the following steps:

1. Activate the tank floor (the annular ring joined to the bottom strake) and allow it to make contact with the top of the foundation, force the foundation to move to the prescribed initial profile (if applicable).
2. First strake sub-model: allow the strake to contact the annular ring but with radial translations of nodes at the bottom of the strake constrained (this constraint represents radial guides fixed to the annular ring, with the purpose of controlling the shape of the bottom edge of the tank). Allow the strake to attain its natural shape under the action of gravity.
3. Lift the annular ring to the calculated bottom edge of the first strake and fix together (this models the construction methodology followed and was achieved via a Python script.)
4. Apply the incremental foundation differential settlement
5. Strake submodel: calculate the assembled shape of the next strake based on shape of the top of the current strake. Update the next strake nodal coordinates in the main model.
6. Restart the main analysis, activate the next strake.
7. Deactivate the wind girder on the current strake and activate it on the next strake (from strake 3 upwards).
8. Continue the process from step 4 until the wind girder is activated on strake 10.

After the activation or deactivation of any part, the main analysis is allowed to run until the structure reaches equilibrium.

Analysis results

The final deformed shape of the tank is shown for the two analysis cases in Figs 4 to 7. In all plots, a deformation scale factor of 20 is used. In Figs 4 and 5, fringes of radial deformation are shown and in Figs 6 and 7, fringes of vertical deformation.

The component deviations from the ideal geometry at the top of the tank for the two analysis cases are compared in Fig. 8. The deviation magnitudes are generally lower for case B, where the foundation is initially level. In particular, much larger radial deviations are seen for case A. Tangential deviations are of similar magnitude for the two cases, as well as vertical deviations. The reason for similar vertical deviations in the two sets of results is that the first strake has sufficient vertical stiffness to span across severe dips in the foundation profile, as shown in Fig. 9. The amount of vertical deviation is thus dictated by spanning of the tank wall across the dip in the foundation, more than the foundation profile itself.

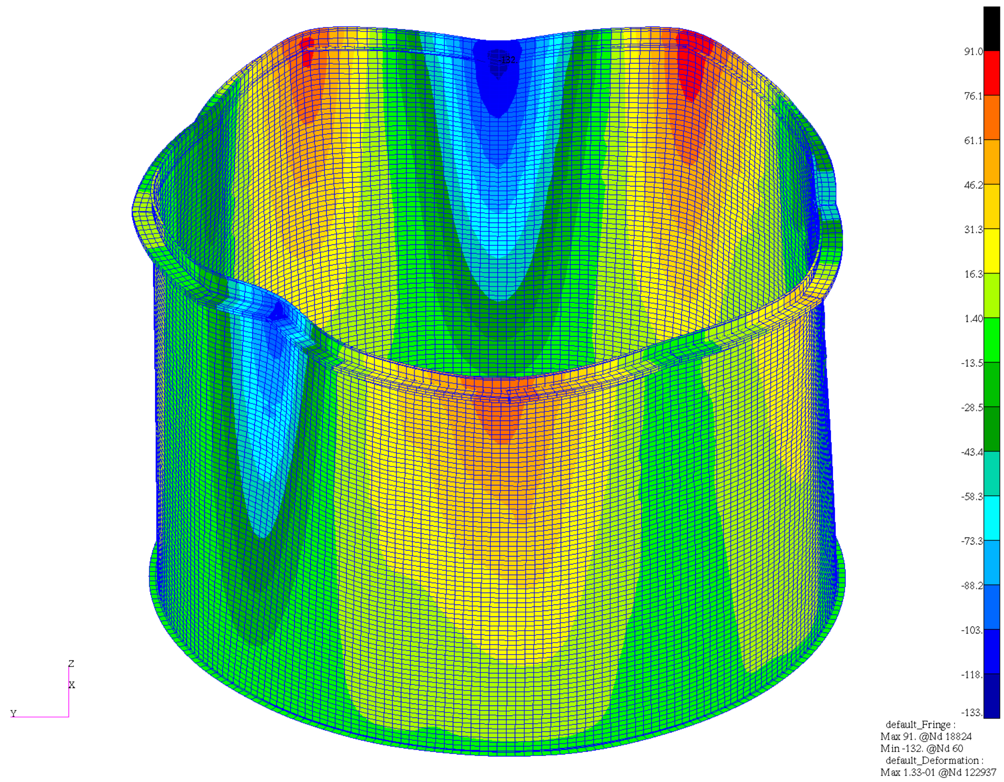

Figure 5: Analysis A - radial deflection [mm]

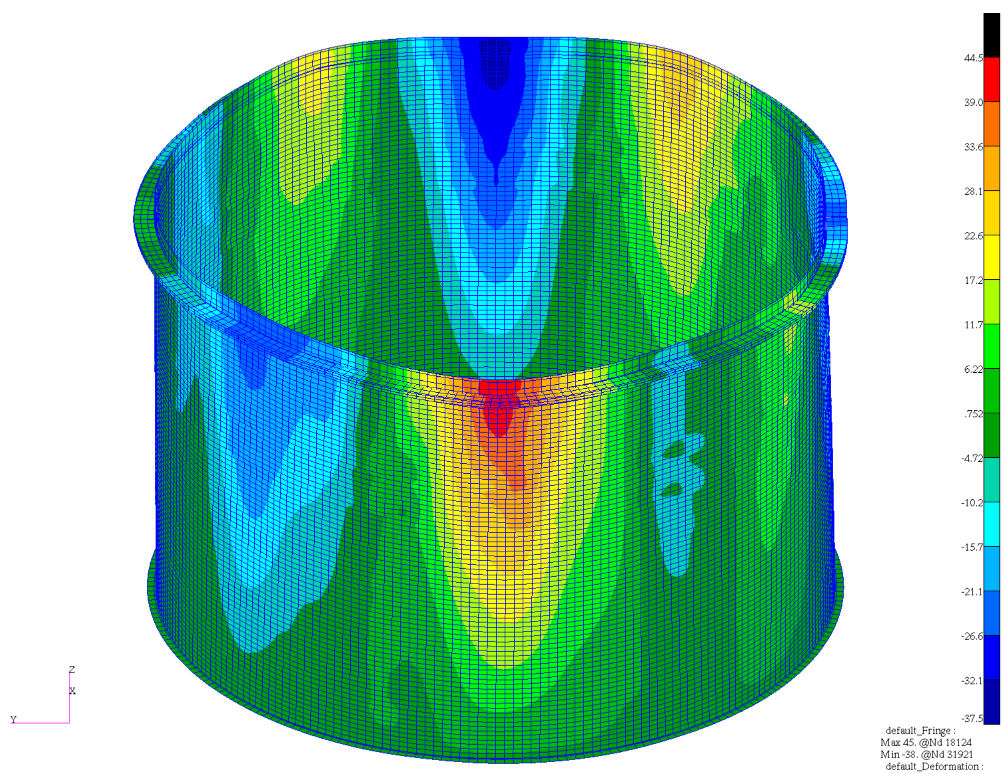

Figure 4: Analysis B - radial deflection [mm]

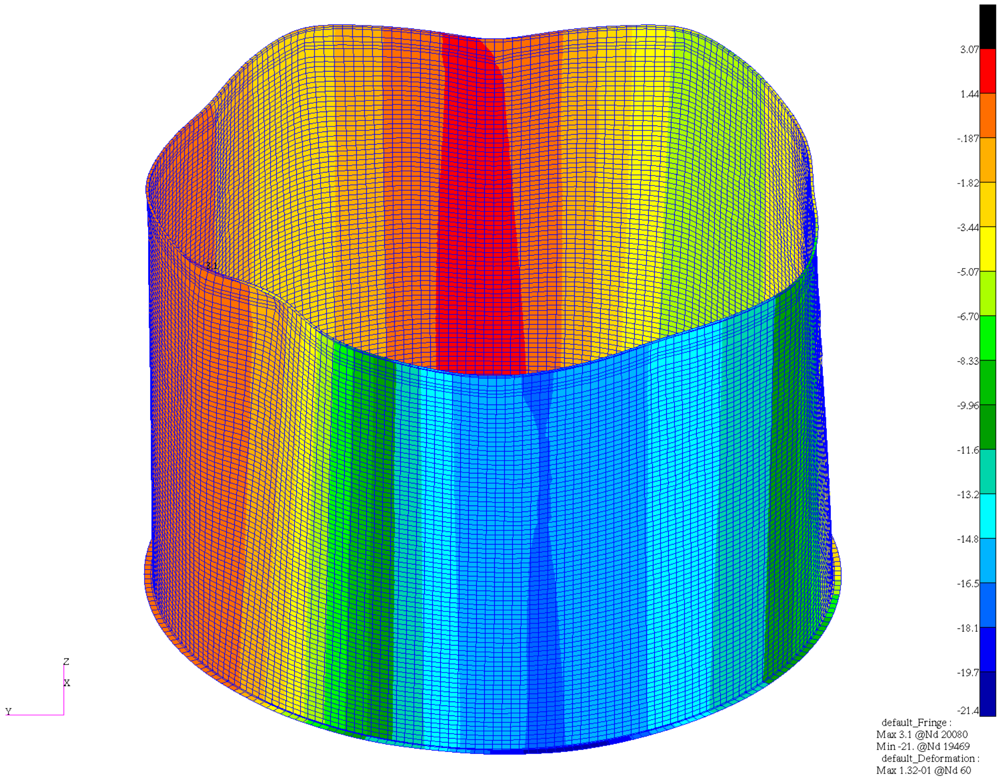

Figure 6: Analysis A - vertical deflection [mm]

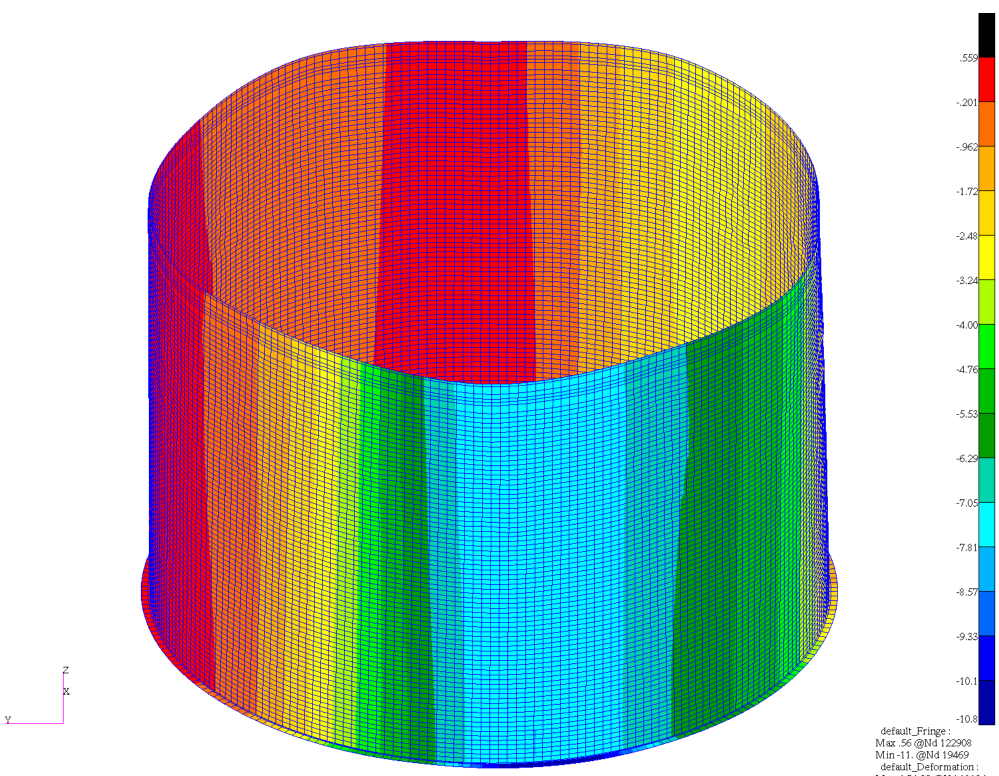

Figure 7: Analysis B - vertical deflection [mm]

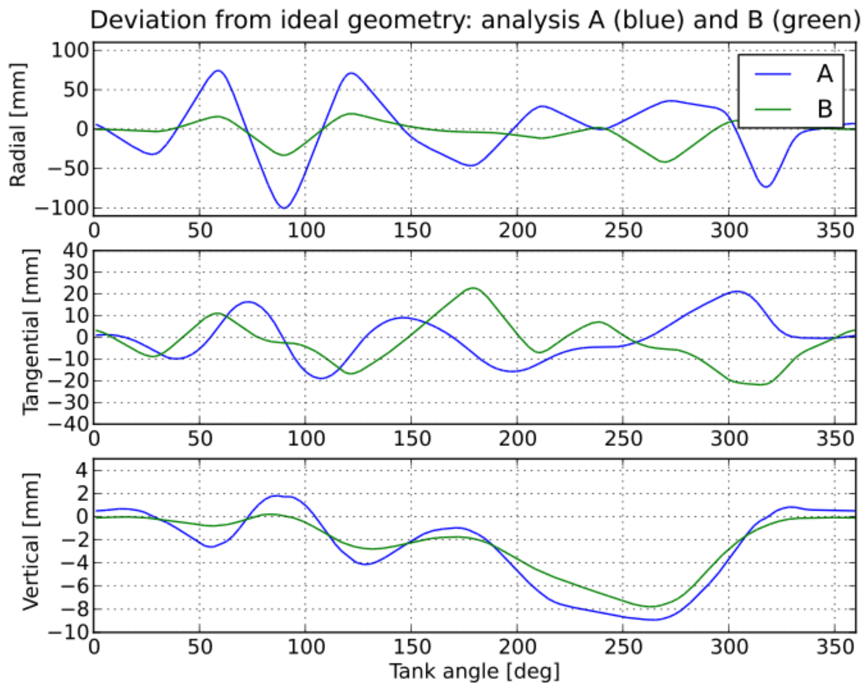

Figure 9: Component deviations from ideal shape at top of tank

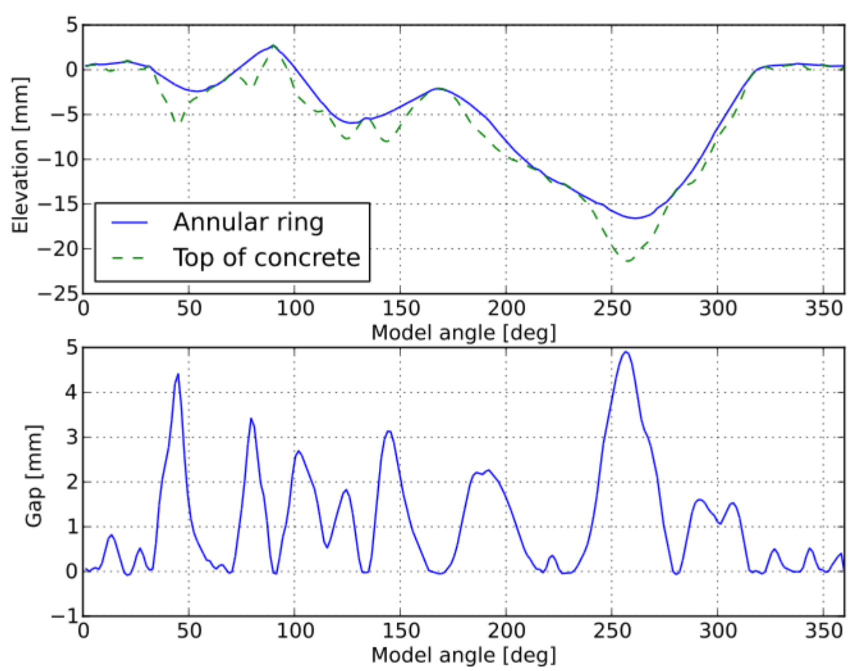

Figure 8: Analysis A – contact between the annular ring (bottom of strake 1) and the top surface of the concrete foundation

Conclusion

An analysis methodology for modelling the bottom-up construction of an API 650 tank on a continuously settling foundation has been developed. In addition to foundation differential settlement, the effect of an initially non-flat foundation is modelled. Analyses were enabled by the LS-DYNA staged construction functionality, and for each tank strake, the assembled strake shape was calculated by means of a sub-model where the strake shape was forced to comply with the strake below it. Python scripts were used to transfer data between the main model and sub-models.

Modeling of the construction process enabled quantification of the effect of foundation differential settlement and initial foundation irregularity on the final constructed shape of the tank, while also including the effect of the wind girder in its temporary locations.

References

1. Welded Steel Tanks for Oil Storage, API Standard 650, American Petroleum Institute, 2005.