ADVANCED BUCKLING ANALYSIS OF A REACTOR TANK SUPPORTED BY FOUR LEGS

MICHAEL TURNQUIST, M.SC., Senior Consulting Engineer at Quest Integrity
ADvanced buckling analysis of a reactor tank supported by four legs

By: Michael Turnquist, M.Sc., Senior Consulting Engineer at Quest Integrity

Editor’s Note: This case study is a good lesson learned about design. For mechanical engineers it is a good reminder. The analogies will help anyone understand what happened and why. It also helps inspectors appreciate the need for integrity of items like stiffener rings, gussets, etc. and why it is important to pay attention to their condition. They often corrode, crack, etc.

Introduction

The failure of a 15-foot diameter reactor tank supported by four legs was analyzed using advanced engineering assessment methodology. At the time of the incident, the cause of the disruption was unknown. However, the failure was believed to be caused by buckling of the support legs. Shortly after the reported failure, another tank of identical design was immediately taken out of service so that an investigation could be performed.

To determine the cause of the failure, an analysis was executed to determine the buckling capacity of the as-built tank design. Buckling is a complex phenomenon and requires the application of advanced finite element analysis (FEA) techniques. Several complexities were involved in simulating the event of buckling collapse, including precise modeling of geometric imperfections based on specific fabrication tolerances, elastic-plastic material behavior simulation, and the use of non-linear and explicit finite element solving techniques.

An independent design review found that the tanks had been constructed without a compression ring at the bottom cone floor-to-shell knuckle weld, which is a violation of API 620 [1]. The results of the buckling analysis indicated that the failure was likely due to excessive compressive stress at the floor-to-shell knuckle weld. This article will outline the engineering and assessment methodologies applied during the analysis of this tank failure.

Background

The number one concern for operators in the oil and gas industry is avoiding equipment failures. This is true from both a safety standpoint, as well as an economic standpoint. Common causes of failure include, but are not limited to, improper operation, inadequate design, and unexpected loading conditions. No matter the consequence, if a failure occurs, it is vital for the operator to understand the cause so that lessons can be learned and future failures can be prevented.

Basic information for this tank is summarized in Table 1.

A sister tank of identical design had not yet experienced a failure, but was taken out of service immediately while an investigation was carried out. This investigation included an engineering analysis to answer the following questions:

• What was the cause of failure?

Table 1. Basic Tank Information

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>15 feet</td>
</tr>
<tr>
<td>Height (shell)</td>
<td>16 feet</td>
</tr>
<tr>
<td>Thickness (shell)</td>
<td>0.25 inches</td>
</tr>
<tr>
<td>Thickness (bottom cone)</td>
<td>0.25 inches</td>
</tr>
<tr>
<td>Support leg W-shape</td>
<td>W10x26</td>
</tr>
<tr>
<td>Max fill height</td>
<td>15 feet</td>
</tr>
<tr>
<td>Product SG</td>
<td>1.54</td>
</tr>
<tr>
<td>Operating temperature</td>
<td>225 °F</td>
</tr>
<tr>
<td>Material (all components)</td>
<td>ASTM A36</td>
</tr>
<tr>
<td>Tank weight (empty)</td>
<td>23,000 lbs</td>
</tr>
</tbody>
</table>

• Is the original design adequate?

• What should be done with the sister tank?

Finite Element Analysis

Finite element analysis (FEA) was performed in order to simulate the failure. The results from the FEA would be used to determine which components experienced the most severe stresses and strains, and subsequently which components were expected to fail first.

The finite element model (FEM) was constructed using the Abaqus [2] CAE finite element modeling program and solved using the Abaqus [2] 2016 explicit FEA solver. The FEM was constructed using linear shell elements (S4R). Non-linear geometry effects were accounted for and elastic-plastic material properties were used for the analysis. Figure 1 shows the mesh used for the FEA.

Material Properties

In order to perform the elastic-plastic analysis, a stress-strain...
curve needed to be defined. A stress-strain curve for ASTM A36 was obtained from tensile tests performed by University of New Mexico Civil Engineering Department [3]. This stress-strain curve is shown in Figure 2.

**Loads and Boundary Conditions**

The loads considered for the FEA included gravity loads (including an additional non-structural mass to account for the weight of additional internal and external components) and the hydrostatic pressure from the product. The loads on the tank were increased by adjusting load factor. Using this approach, the load factor where failure was predicted to occur could be identified.

The bottom of each support leg was supported by a load cell. The load cell was designed to fail once its allowable lateral load was exceeded. In order to simplify this boundary condition, two cases were analyzed. Case 1 consisted of the bottom of the support columns as fully restrained against lateral movement. Case 2 consisted of the bottom of the support columns as un-restrained against lateral movement. Using this approach, the problem could be bounded since the actual condition was somewhere between these two cases. These two cases are summarized in Figure 3.

**Geometric Imperfections**

In order to effectively simulate a buckling failure, small geometric imperfections (decrease the buckling capacity of a structure) need to be considered. Consider the classic “soda-can” example: If you take an empty soda can, place it vertically on a table, and put a weight on the top of it, the can will be able to support the weight. However, as soon as you touch the side/center of the can with a pencil (do not use your finger!), the can collapses. This is because by touching the side of the can with the pencil, a small imperfection was introduced into the can causing a decrease in buckling capacity.

Geometric imperfections were included in the support leg geometry based on fabrication tolerances taken from ASTM A6 “General Requirements for Rolled Structural Steel Bars, Plates, Shapes, and Sheet Piling” [4]. The support leg geometry was adjusted based on fabrication tolerances for web out-of-plumbness and overall camber. This is demonstrated in Figure 4.
RESULTS

The FEA was run by increasing the load factor until buckling collapse of the structure was observed. Both boundary conditions cases experienced similar failure modes. The structural behavior of the tank was dominated by “catenary action” of the cone bottom which caused the shell-to-bottom weld area to compress like a “compression ring.” The structural response (or the load carrying behavior) of the tank is summarized in Figure 5. This is a plot of vertical reaction force (measured at the floor level) of the entire tank vs. vertical displacement of the tank. Key failure events are indicated on the plot.

Catenary action is best explained using an analogy of a clothes line supported atop two posts. When heavy wet laundry is hung, the clothes line will pull the two poles toward one another. Similarly, the cone bottom sagged and pulled the shell inward when hydrostatic pressure was applied. This can be seen in the contour plot of compressive circumferential stress shown in Figure 6. As the shell was pulled in, a ring of material at and adjacent to the shell-to-bottom weld was compressed, forming a “compression ring” zone. This plot corresponds to when the load factor was at 0.2, or 20% of the expected design load. Already the maximum allowable compressive stress had been exceeded at this location.

Figure 6. Compressive circumferential stress in the compression ring zone at LF = 0.2. Cross sectional view.
Once the load factor was increased to just above the expected design conditions (LF ~ 1.1), bulging of the bottom cone to shell knuckle was observed. At this point, buckling was initiated. As a result of this, high tensile stresses began to form near the knuckle, leading to a risk of tensile rupture.

A plot of load factor versus compressive plastic strain is shown in Figure 7. This plot further supports the observation that the knuckle was exposed to very severe compressive stress, resulting in the material to yield in compression. Strain-hardening behavior was observed around LF~1.1 (the same point when buckling was initiating).

The tank reached its peak vertical load capacity at LF=2.06, indicating that buckling of the support legs had initiated. However, by the time this point was reached, the stresses in the bottom knuckle were to the point where a failure would have occurred before buckling of the support legs would initiate.

Results Summary
To summarize the collapse behavior of the tank, catenary action caused the shell to pull inward, developing compressive stress at the knuckle. The compressive stress exceeded the allowable value when the Load Factor = 0.2. As the load factor was increased to about 1.1, excessive compressive stress in the knuckle caused bulging/buckling. This behavior caused tensile stresses to develop, and a risk of tensile rupture was observed.

The peak load capacity of support columns was reached at LF=2.06, indicating that buckling of the support legs had initiated. However, the analysis suggested that the knuckle failed before the support legs buckled, although in real time these events likely happened very quickly.

In addition to the analysis described above, an additional design review found that the vessel was designed without a compression ring. It was determined that the sister tank was also missing a compression ring, indicating that a similar failure would likely occur if the sister tank was returned to service. It was recommended that a properly designed compression ring be installed on the sister tank before returning it to service. Additionally, a thorough inspection was carried out to ensure that no other components were damaged as a result of the tank being operated with an inadequate design.

CONCLUSIONS
This engineering analysis served as an important reminder of what can happen when a piece of equipment is improperly designed. In this case, the tank had been constructed without a compressing ring at the bottom cone floor-to-shell knuckle weld, which is a violation of API 620 [1]. The results of the buckling analysis reinforced just how important this component was to the design, as the FEA results indicated that the failure was likely due to excessive compressive stress at the floor-to-shell knuckle weld and not due to buckling of the support legs as originally believed.

For more information on this subject or the author, please email us at inquiries@inspectioneering.com.

REFERENCES
Manage the risk of unplanned downtime, loss of production or a catastrophic failure in your fired heaters.

Quest Integrity’s Furnace Tube Inspection System (FTIS™) is a globally proven technology providing 100% inspection coverage of your serpentine coils, compliant with the API-573 Fired Heater Inspection guidelines. The FTIS inspection results are processed with our LifeQuest™ Heater engineering software, providing a comprehensive fitness-for-service and remaining life assessment compliant with the API-579 Standard. Quest Integrity delivers a complete solution that helps transfer your integrity and maintenance risk into reliability.

- Pitting (interior or exterior of pipe)
- Corrosion (interior or exterior of pipe)
- Erosion and flow assisted wear
- Denting and ovality
- Bulging and swelling
- Coke and scale build-up

Get the information you need to confidently make decisions on your fired heaters with plug headers.

To learn more about FTIS, visit us at the link below.
**CONTRIBUTING AUTHOR**

**MICHAEL TURNQUIST, M.SC.**

Michael Turnquist is a Senior Consulting Engineer with Quest Integrity’s Advanced Engineering group in Boulder, Colorado. His project experience includes various types of engineering fitness-for-service assessments, with the majority of that experience in the application of finite element analysis (FEA) and fracture mechanics to the assessment of asset integrity. Michael has a BS degree in Civil Engineering and a MS degree in Structural Engineering from the University of Colorado, Boulder.