Application of Finite Element Methods for Evaluating Hazards to Equipment

D. Powell, Ph.D., Senior Consultant, & D. Bogosian, P.E., Senior Principal Engineer, Baker Engineering & Risk Consultants, 360 N. Pacific Coast Hwy., Suite 1090, El Segundo, CA 90245, USA

Because of their inherent complexity, industrial equipment such as pressure vessels or storage tanks are not usually amenable to analysis using simplified analytical techniques. Fortunately, improvements in computer efficiency and modeling software and the widespread availability of commercial finite element codes have made finite element analysis (FEA) a viable and potent tool for analyzing a variety of hazards to which non-structural items may be subjected.

This presentation will examine three real-world case studies in which the FEA program LS-DYNA was used to evaluate the potential damage to equipment for a given incident and provide necessary information for owners and operators to make key decisions regarding remediation or retrofits.

In our first case study, FEA was used to predict the response of an isotainer containing hazardous materials when accidentally dropped onto another isotainer. This incident represented the uncontrolled descent of the isotainer while being lowered by a crane during stacking operations. The simulation predicted the deformation of the frame and inner shell of the two isotainers and predicted that the plastic strain in the shell would not exceed the minimum threshold for material failure. However, a valve was sheared off during the incident, resulting in a prediction of loss of containment (LOC), which was then factored into a refined quantitative risk assessment.

The second study examined the response of several large storage tanks (both cylindrical and spherical) to blast loading from an adjacent accidental event. While the design of some of the tanks was sufficient to withstand the blast loads, connection failure at the base of one tank was predicted in the model. From this high-fidelity model, the predicted damage and cost of retrofitting the storage tanks could be weighed against the potential business interruption to assist the client in making an informed decision.

Finally, we were asked to evaluate the secondary containment for two 30,000-ton ammonia tanks to address a concern whether a catastrophic failure of the tanks could result in the immediate release of the contents and overwhelm an earthen berm and reinforced concrete retaining wall intended to contain a more gradual release. In this case, LS-DYNA was used to run a computational fluid dynamics (CFD) model of the fluid's motion under gravity. The resulting pressure it would apply on the earthen berm and concrete containment wall were then extracted from the simulation. Those pressures were used as inputs to an analytical model and an SDOF (single degree of freedom) model to predict the response of the berm and wall, respectively. That study determined that the berm would survive; however, a portion of the concrete wall was predicted to fail and release the liquid ammonia into the surrounding area. As a result, the client could focus their efforts on upgrades to the retaining wall and not the earthen berm.

These case studies illustrate the benefits that high fidelity finite element models can provide — more accurate prediction of the response of complex systems; evaluation of risks to the business operation; and, in cases where deficiencies are identified, pinpointing the regions of failure for more targeted and therefore less costly retrofits. As numerical tools continue to improve, the cost associated with using them decreases, making their use in safety analysis and decision making much more accessible and cost effective.

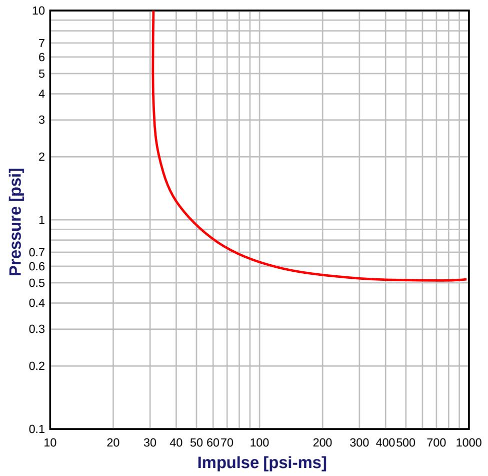
Introduction

The goal of structural analysis is to predict how a structure or a piece of industrial equipment responds to a set of applied loads. For typical service loads, the analysis may be done statically, by applying a constant load and computing a static response with no time dependence. For hazardous events, such as an explosion, a drop, or an impact, the event is transient and the loads are highly time dependent. Consequently, a dynamic method is needed.

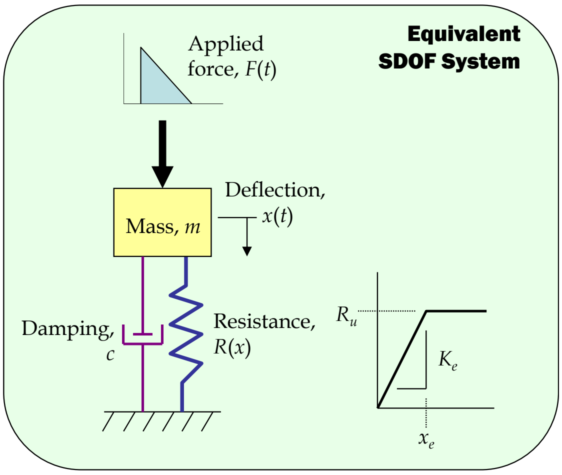
There are a number of commonly used techniques for such analyses. When available, a Pressure-impulse (P-i) curve (Figure 1a) is a convenient and simple way of determining whether a particular applied load produces a response that exceeds a given criterion. The curve represents a given level of response or damage, and the applied load is represented as a point placed at the corresponding pressure and impulse values for that load. Points that fall below the curve indicate a response that meets the criterion, while points above the curve indicate a scenario that will exceed the criterion. While quick and simple to apply, P-i curves are limited in the information they provide about the structure and its response. For example, P-i curves are typically limited to very simple load histories (e.g., triangular) and basic structural components (e.g. a slab or a beam). Curves for more complex structures require either an extensive set of experiments or a more complex form of analysis to first generate the P-i curve.

A single degree of freedom (SDOF) model (Figure 1b) is another technique which has the advantage of allowing the user to run any arbitrary load history, and can represent a wide variety of components [Biggs 1964]. It also provides more detailed information on the time-dependent response, such as peak deflection. However, it is still typically limited in the complexity of the structures to which it can be applied to (e.g. beam, column, or reinforced concrete wall) and requires the user to make assumptions about the way in which the structure will respond before running the calculations. Its output is also limited to a single deflection, and it is unable to account for interactions between multiple components or any boundary conditions other than the most idealized (fixed, simple).

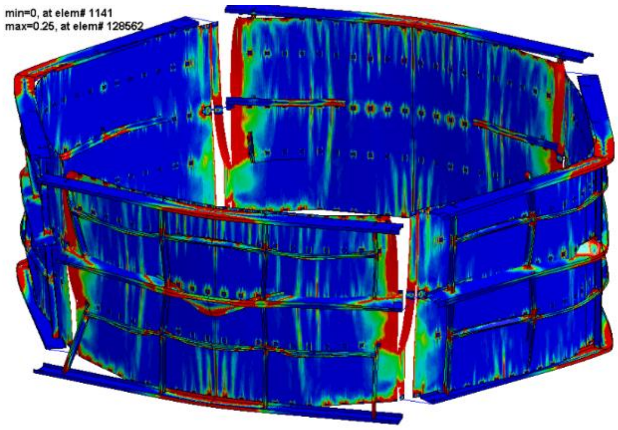
A finite element analysis (FEA) (Figure 1c) is a substantially more detailed representation of both the structure and the loading, thereby allowing for the highest level of fidelity. In this method, the structure is subdivided into a large, but 'finite', number of elements (typically numbering in the hundreds of thousands to millions, depending on available computer resources) and a similarly large number of nodes. At each time step, loads are applied, stresses and strains are calculated within each element, and the equations of motion are solved at each node. This approach produces the most information about the response of the structure and can define locations of high stress or areas of failure. It can also include higher order effects like contact, impact, material failure, load redistribution, large displacement effects, and thermal effects. Furthermore, the complexity of the FEA is limited only by the available resources of time and budget.



(a) Pressure-impulse curve



(b) Single degree of freedom



(c) Finite element analysis

Figure 1: Examples of common analysis methods for evaluating the transient dynamic response of a structure.

Simpler approaches like P-i curves and SDOF require a number of assumptions; should a problem fit those idealizations, the speed and low-cost of those approaches make them perfectly reasonable options. But, when a problem no longer conforms to those assumptions, these simple models need to be replaced by a more complex one capable of representing the features missing from the simpler model. In this work, we will examine three case studies from BakerRisk's recent projects in which FEA was used to evaluate the response of equipment to a variety of hazards. For each example we will discuss why FEA was necessary, how it was used, and the benefits that analysis provided to the client. Although FEA is typically considered a tool for structural analysis, we will demonstrate its value and power in the analysis of equipment subjected to high-intensity, dynamic loading.

The case studies presented below were all performed with LS-DYNA, an explicit solution, large deformation, dynamic finite element analysis code [LSTC 2018]. The meshes were generated using LS-DYNA's pre and post-processing software, LS-PrePost, in conjunction with specialized tools developed by BakerRisk to expedite the mesh generation process.

Application of Finite Element Modelling Case Studies

Case Study 1: Isotainer Drop Scenarios

Anhydrous hydrogen fluoride (AHF) is routinely transported in isotainers by truck and by ship. The isotainers consist of a steel tank, cylindrically shaped, contained within a cradle or support frame constructed of structural steel sections (Figure 2). Concerns were raised regarding potential loss of containment (LOC) if the isotainer were dropped during crane operations. In order to quantify this risk, high-fidelity modelling was required to determine whether any of a variety of drop scenarios would result in LOC. The complexity of the isotainer design with interactions between the shell, frame, and fittings during the impact precluded the use of simplified models and indicated the necessity for FEA.

Figure 2: Example of a two-valve isotainer.

A finite element model of the isotainer (Figure 3) was created. Shell elements were used for the frame, which consisted primarily of steel hollow structural section (HSS) members along with a smaller number of channels and plates, as well as for the tank shell containing the liquid AHF. The AHF within the isotainer was also included in the model, represented using solid elements. Typical element size, shown in the close-up image, was 40 mm square for the steel components and 40 mm cubes for the internal fluid. The fittings, consisting of the liquid and vapour valves, manway, and pressure release valve, were also discretely modelled. While deformation and even local failure of the surrounding frame would be acceptable, any failures predicted in the tank shell or fittings represented potential sources for a LOC. The bolted connections between the liquid and vapour valves and the shell were discretely modelled as well. Numerous contacts were defined for the model so interactions between the frame as it deformed and the tank shell would be represented in the model.

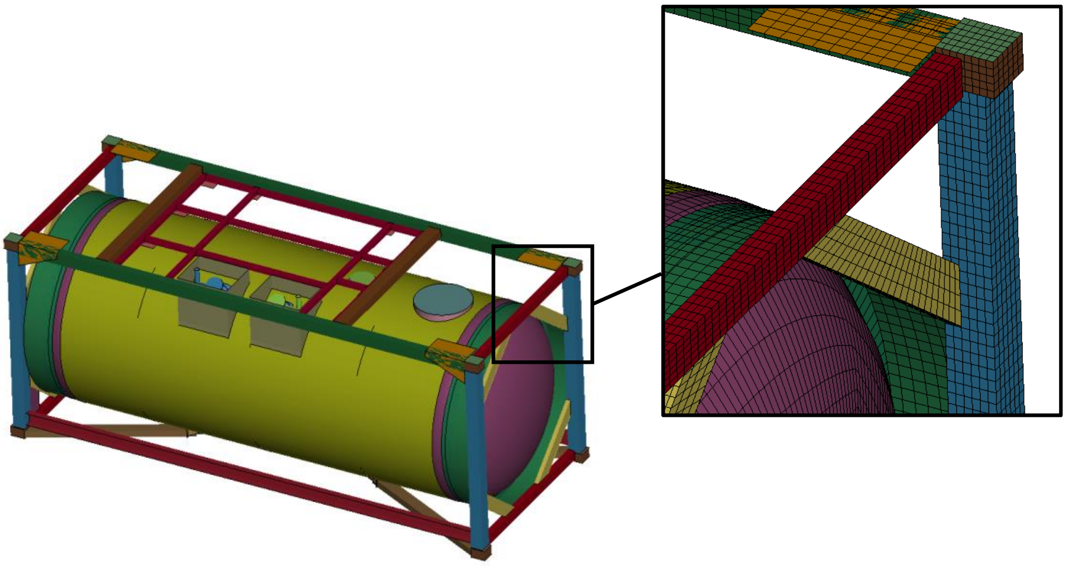
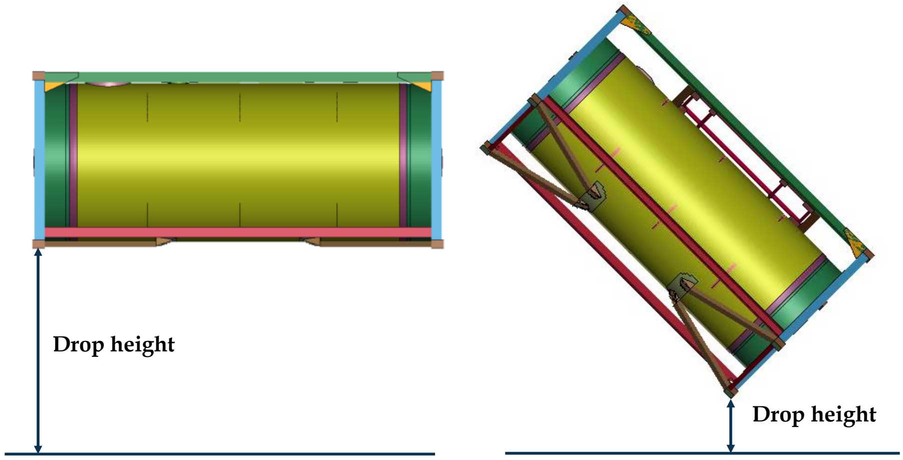


Figure 3: Finite element model of isotainer with enlarged image showing mesh size.

A number of different accident scenarios were evaluated within this study, however here we will focus on two of the drop scenarios. The first involved a drop onto a flat, non-responding (rigid) ground plane (Figure 4). Following that, we examined the impact of one isotainer falling onto another (Figure 5). The placement of the upper isotainer is shifted off-centre and the orientation was rotated by 30° in order to represent a more severe interaction than a perfectly aligned impact.



(a) Flat drop (b) Angled drop

Figure 4: Definition of drop height for both isotainer orientations evaluated.

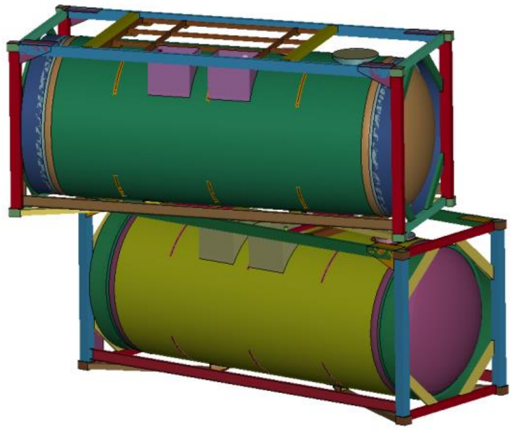
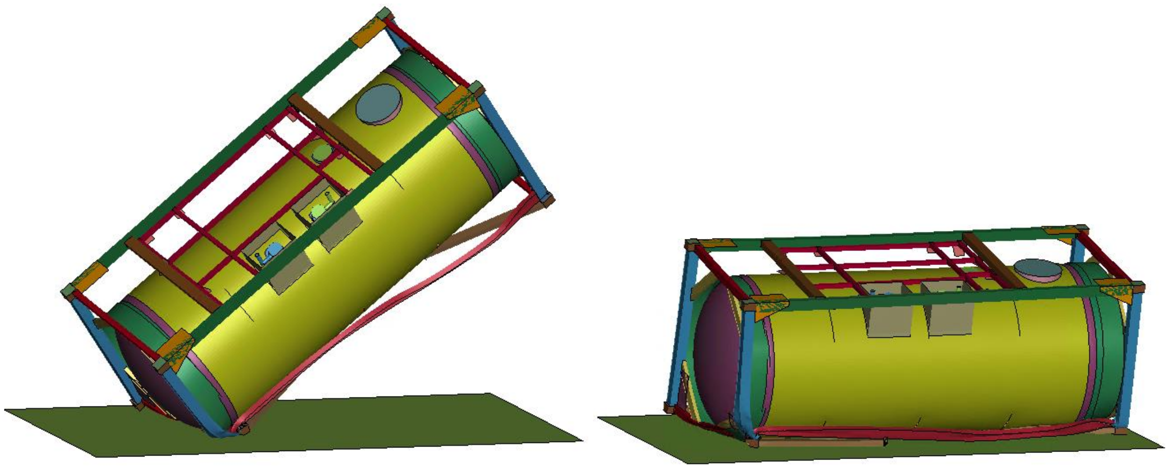


Figure 5: Setup for isotainer impacting a second isotainer during stacking.

An example result for a 2-m angled crane drop onto a flat surface is shown in Figure 6. The initial impact results in a crushing of the lower corner of the isotainer frame until the main shell comes into contact with the ground (Figure 6a) and the downward velocity is slowed, approaching zero. However, the back end of the isotainer then begins to rotate downward, picking up speed until it finally impacts the ground (Figure 6b). This second impact results in further damage to the frame, pressing a portion of the diagonal supports into the shell. While there is significant damage and material failure in the surrounding frame, the plastic strain within the shell does not exceed the minimum specification for failure and thus no LOC is predicted for this drop scenario.



(a) Deformation at 180 ms after initial impact (b) Final deformation at 500 ms

Figure 6: Response of isotainer for a 2-m angled drop.

An example of an isotainer impacting another isotainer is shown in Figure 7. This is a more complex interaction, with first an initial impact of the upper isotainer onto the top of the lower isotainer. During this impact, the frames of the two isotainers press into each other and deform. Following that, the upper isotainer begins to slide over the first and tumble towards the ground, creating a second impact event. During the initial impact and subsequent sliding of one isotainer over the other, the liquid valve of the lower isotainer is sheared off, which was assumed to result in a LOC. The shell of the upper isotainer shows concentrated regions of plastic strain where it comes in contact with the corner of the lower isotainer's frame and where it is pressed into the lower isotainer's liquid valve. However, in both cases, the plastic strain remains below the minimum specification for material failure.

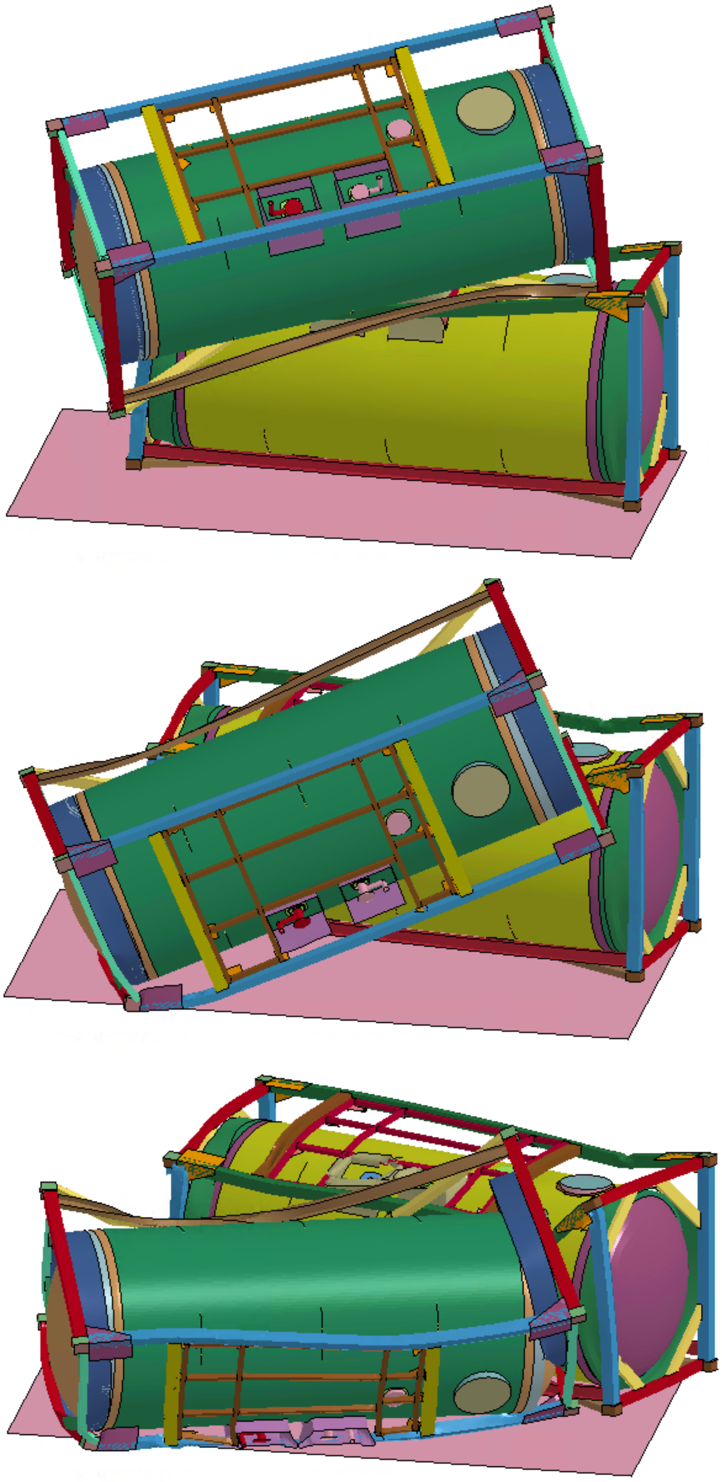


Figure 7: Impact of one isotainer onto another.

Analyses were run for drops of different heights, using two different isotainer designs. For each analysis, a maximum plastic strain in the shell of the tank was recorded. Since these results were desired for use in a quantitative risk assessment (QRA), a probabilistic risk value was desired rather than a binary determination of pass/fail. To associate plastic strain with risk of

LOC, consideration was given to the range of tensile strain at failure for the steel of the shell. At the minimum specification value (21%), we deemed it to be zero. Based on various experimental studies on this material, a value of 28% was deemed to represent the average material response treated, conservatively, as certain failure or tearing, and was thus associated with a probability of LOC of 1.0. In between, a linear increase in the probability of failure was assumed. This relationship is plotted in

Figure 8. Combining that function with the plastic strain results of the FEA produced the final failure probability relationships shown in Figure 9.

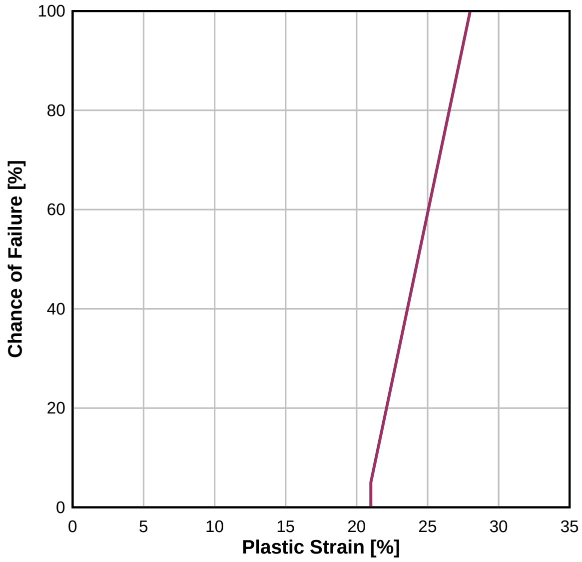


Figure 8: Relationship between plastic strain and probability of failure for ASME SA 516 gr. 70 steel.

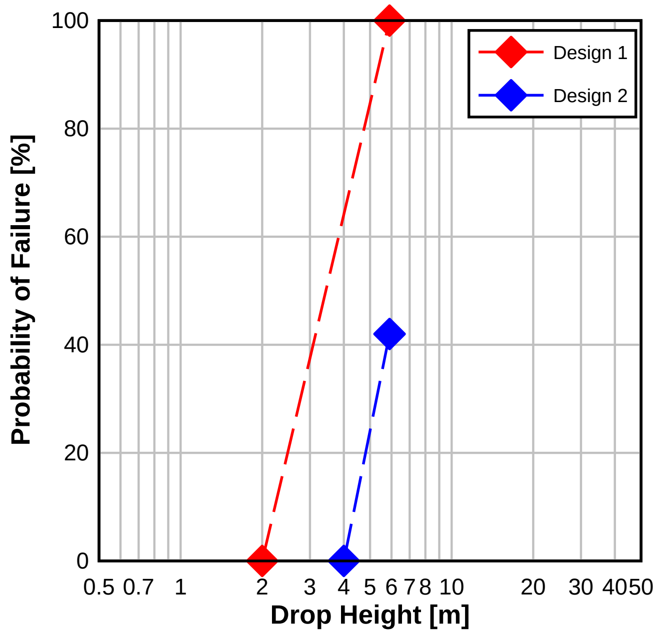


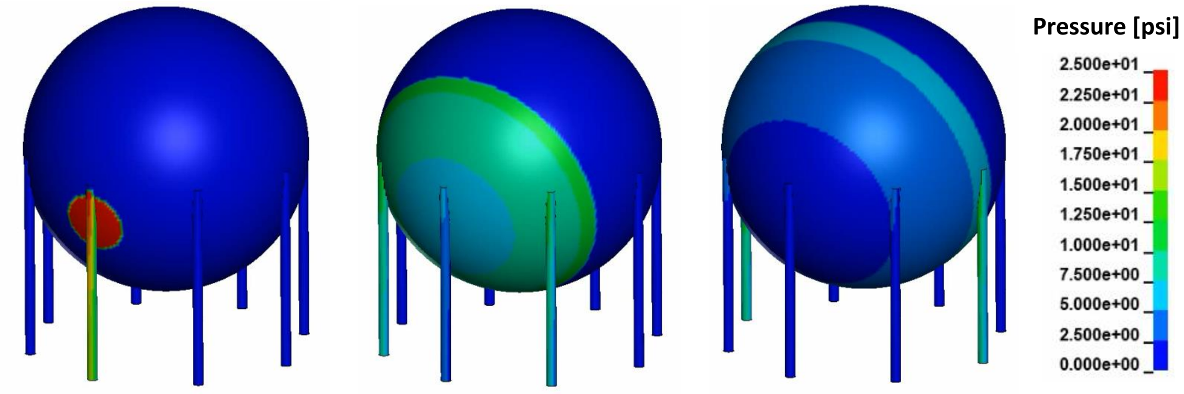
Figure 9: Probability of failure (LOC) as a function of drop height for two isotainer designs.

This study provides an example of FEA's ability to represent highly complex geometries and assemblies to a high degree of detail and fidelity, all of which has a bearing on the analytical outcome. The shearing off of a valve or the indentation of a tank due to contact with an angle stiffener are realistic causes of LOC and can only be predicted if the model is detailed enough to include such elements. Additionally, by considering magnitudes of plastic strain and correlating them to a probability of material failure in the tank, a probabilistic relationship between drop height and risk of LOC could be credibly established, adding to the credibility of the subsequent QRA results. No such outcomes could have been predicted with simple models.

Case Study 2: Storage Tanks Exposed to Blast Loading

In this second case study, two large tanks (one spherical, one cylindrical) were evaluated for blast loads from a hazard scenario associated with new equipment proposed for addition to the site. Because the tanks had been designed and installed before this potential hazard was known, the tanks were not designed for these loads and thus their response was unknown. The complexity of the tank design as well as the complexity of the interaction between the tank and the blast loading necessitated the use of FEA. The blast loads for each storage tank were defined in terms of peak pressures and impulses (area under the pressure history curve). The pressures and impulses thus identified are free-field or incident values at the centroid of the tank. To represent these blast loads in the numerical model, an equivalent TNT detonation was selected at an appropriate standoff and distance which would replicate the incident pressure and impulse at the centroid of the tank. The blast load histories were calculated using the widely accepted TNT blast model documented in standard manuals [U.S. DoD 2014], [Kingery & Bulmash 1984].

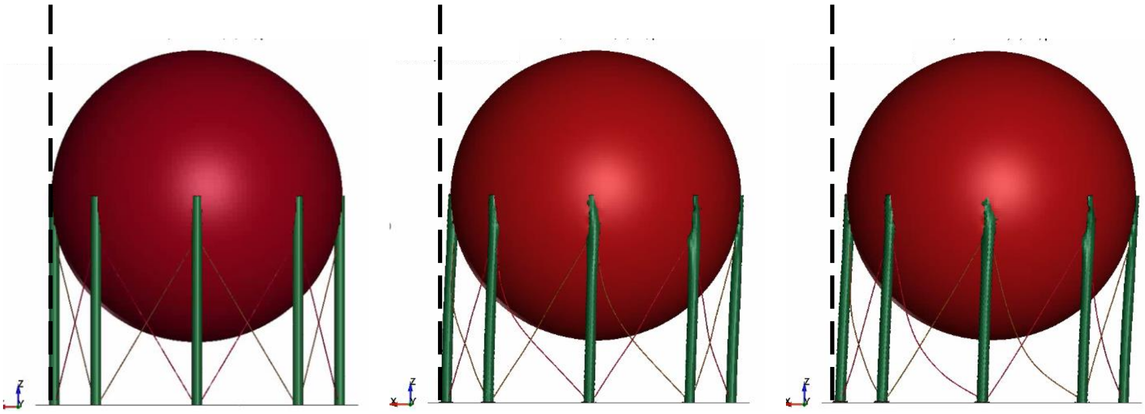
While the blast pressures at various points along the surface of the storage tank may be calculated by hand, some finite element programs, such as LS-DYNA, include tools that will automatically perform these calculations. Given the specified weight of TNT and standoff from the item, a feature in LS-DYNA called LOAD_BLAST_ENHANCED was used to calculate blast pressure at each point along the surface of the tanks. This includes the time of arrival and, where appropriate, the reflection factor for the blast pressure, taking into account the angle of incidence at each point along the surface. Figure 10 illustrates the applied blast pressure on a spherical tank. The calculated incident pressure at the centroid was 7.4 psi, however the applied pressure at the point on the surface facing the charge is much higher, 25 psi, due to both its closer distance to the blast source and the applied reflection factor. As the blast wave reaches the 45° azimuth (Figure 10b) it falls to around 13-14 psi and finally to 7.4 psi at the 90° azimuth. A small part of this drop in magnitude is due to attenuation (i.e., the expansion of the shock as it travels over the 28' radius of the tank), but most of it is due to the reduction in the reflection factor from 0° to 90°.



(a) t = 57 ms (b) t = 69 ms (c) t = 81 ms

Figure 10: Pressure from blast wave sweeping over spherical tank at various times.

The blast loading on the spherical tank produced a slow lateral translation resisted primarily by the legs acting as cantilevers and diagonal cable bracing connecting the legs (Figure 11). Localized plastic strain was predicted at the bottom of the legs, at the base plate, and where the legs joined the spherical shell. However, the maximum plastic strains were 2% or less, well below the rupture level for the material (over 20%). Furthermore, no plastic strain was observed in the steel shell or any of the internal stiffeners. The peak dynamic forces at the base were extracted from the simulation and compared to the capacities of the existing anchors. Although approaching the capacity, the peak shear and tensile forces did satisfy the design criteria and thus the storage tank was concluded to need no further retrofits.



(a) t = 0 ms (b) t = 161 ms (c) t = 327 ms

Figure 11: Dynamic response of spherical tank (deformations exaggerated 4×).

A similar evaluation was undertaken for a cylindrical storage tank at the same facility. However, in this case, the finite element model indicated that the tank's resistance to the blast loads was limited by the capacity of the anchor bolts at the base. The model showed that, at a very early time, all the bolts would fail. As a result, the lower portion of the cylinder deformed significantly, and at later times, the entire cylinder began to translate laterally in a more-or-less rigid body mode as it was pushed by the blast (Figure 12). In the numerical model, no surface imperfections or obstructions were included, so the cylinder was free to slide laterally without impediment; in reality, it would most likely topple.

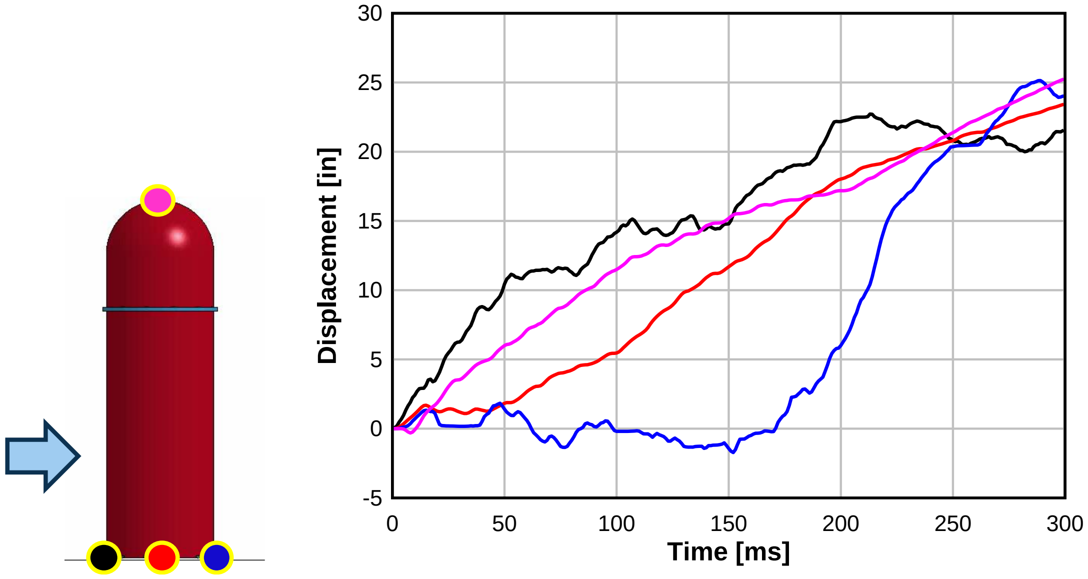


Figure 12: Lateral displacement at bottom and top of cylindrical storage tank.

To understand how the tank itself would respond were the anchor bolts to have sufficient strength, the model was revised such that the bolt capacity was infinite and the bolts could not fail. The resulting tank response was quite benign. Figure 13 shows the lateral displacement at different heights and the peak deformations are at most 2½ inches, but they rapidly return to zero. In other words, the tank responds elastically and incurs no permanent deformation.

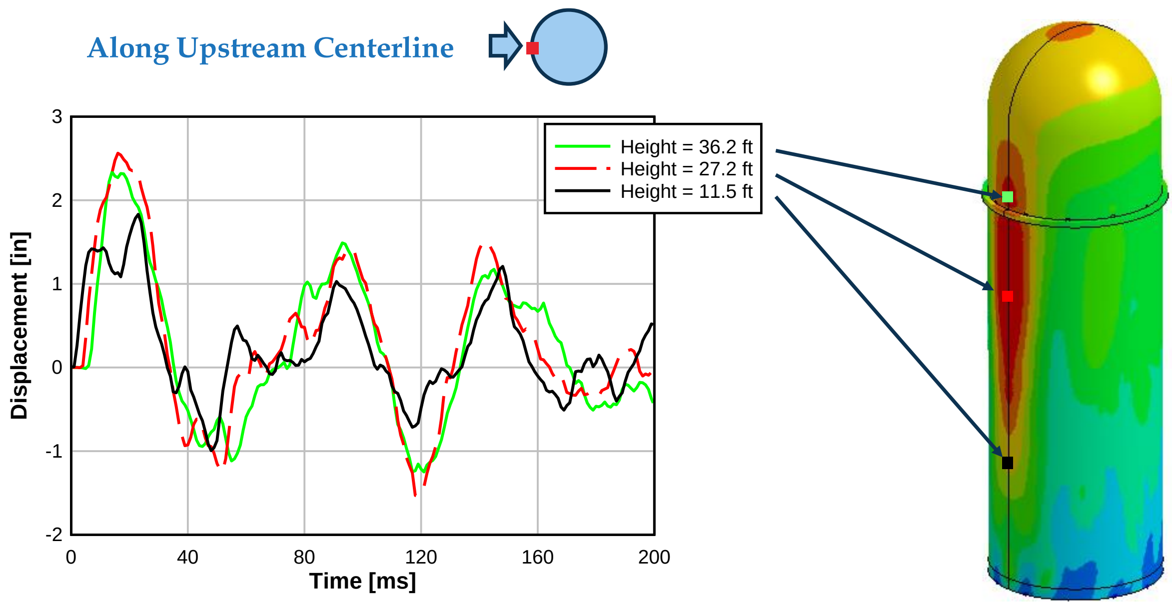


Figure 13: Lateral displacement at various heights after retrofit to base connections.

This study illustrates the ability of FEA to analytically represent large, complex tanks and predict the response of these systems to blast loads that are varying not only temporally but spatially over the surface of the tank. Capturing such gradients of pressure and time is not possible with an SDOF model. The FEA showed that one tank (the spherical one) needed no upgrades to meet requirements, while the other (the cylindrical one) had a weak base connection that would fail readily and potentially cause catastrophic tank overturning. The FEA further showed that, following a relatively low-cost and low-disruption retrofit of the tank's base connections, the structural response of the tank would be benign, thus allowing decision makers to make an informed decision regarding retrofits.

Case Study 3: Secondary Containment Response

This final case study examines the response of the secondary containment measures for a 30,000-ton ammonia tank in the event of a catastrophic failure and LOC. The secondary containment measures consisted of an earthen berm and a steel-reinforced concrete containment wall. Both of these are sufficiently simple geometrically that they can be evaluated using engineering level models (SDOF). However, these approaches require details on the loads applied to the structures due to their interactions with the fluid. Analytical models for the pressure loads may be used in the case of a gradual release but do not exist for an instantaneous release due a catastrophic tank failure. In order to determine those loads, BakerRisk used LS-DYNA to perform a computational fluid dynamics (CFD) analysis (essentially a subset or category of FEA).

An axisymmetric model was created to represent the ammonia within the tank and the ground, including the slope of the berm, as shown at $t = 0$ in Figure 14. The use of an axisymmetric model here is conservative as it predicts the pressures on the berm at its closest distance to the tank and was chosen to control the computational cost and time associated with running this large CFD model. The simulation treats the ground and the berm as rigid while also recording the pressures from the fluid at various points along the berm. At the initiation of the simulation, the fluid ammonia is released (representing a catastrophic failure as though the steel tank has instantaneously vaporized and disappeared) and allowed to respond without any restraints in the horizontal direction. The resulting motion of the fluid, under the effect of gravity, is shown in the remaining panels of Figure 14. As the fluid column collapses, it expands radially and, after impacting the berm, a portion of the fluid ammonia continues to flow up and over it, escaping the secondary containment. However, the primary concern of the client was the structural response of the containment measures and whether they would remain in place to contain the bulk of the fluid released.

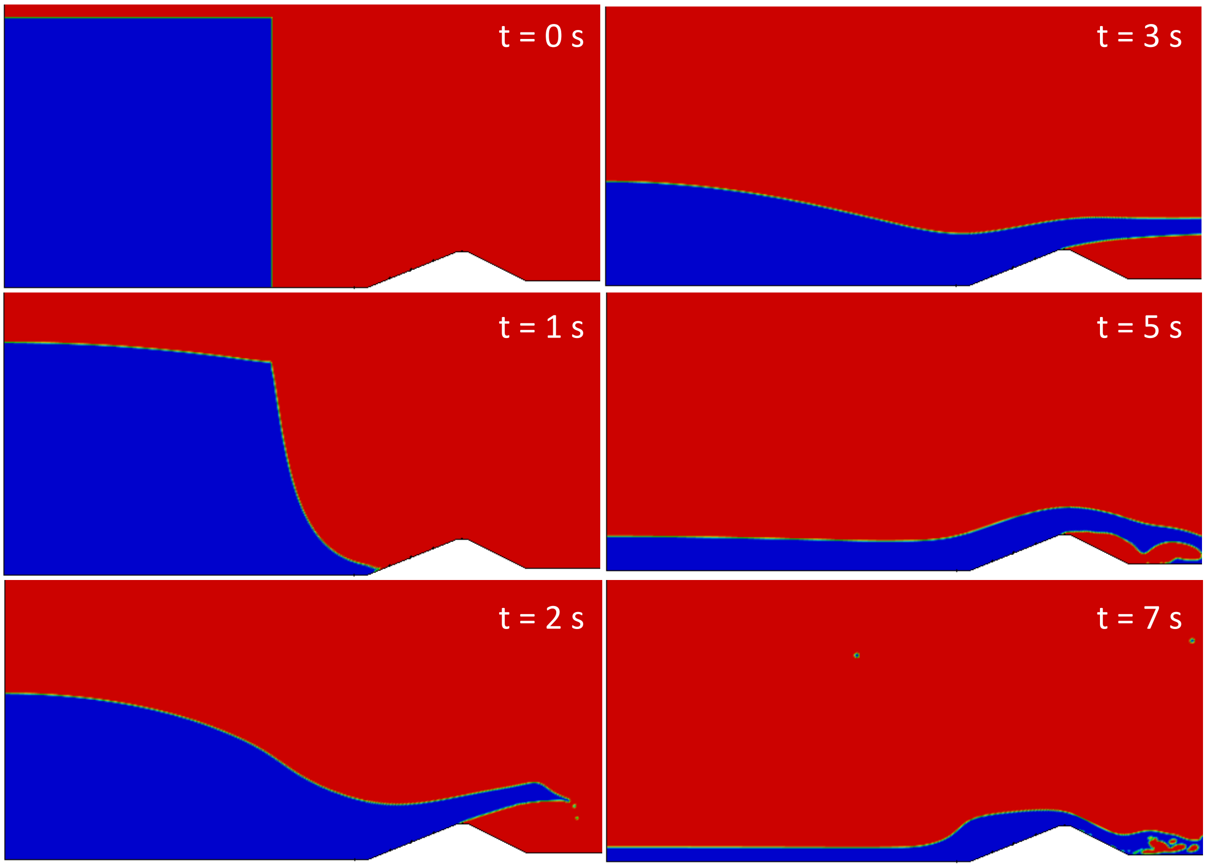


Figure 14: CFD Simulation of ammonia fluid flow over berm.

A similar analysis was performed for a vertical containment wall (Figure 15), which replaces the berm to one side of the tank. In both instances, the pressure histories were recorded at a number of points along the containment measures (Figure 16). For the berm, the simulation predicted peak pressures of 11, 7, and 4 psi as the fluid travelled up the berm. The wall, oriented at 90° to the fluid flow, experienced significantly higher pressures reaching 35 psi at the closest point to the tank. It can be noted that the wall is constructed further from the tank than the berm, hence the later arrival time for the pressures in the second plot.

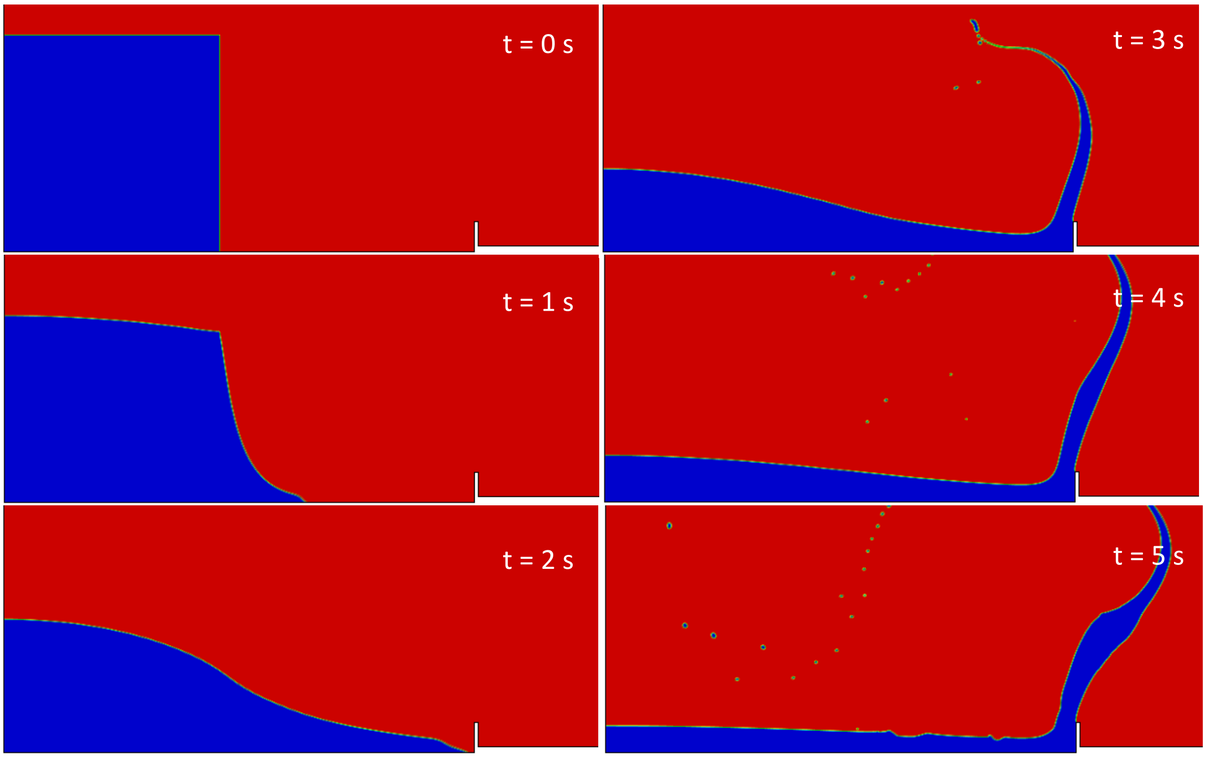


Figure 15: CFD simulation of fluid flow interacting with the containment wall.

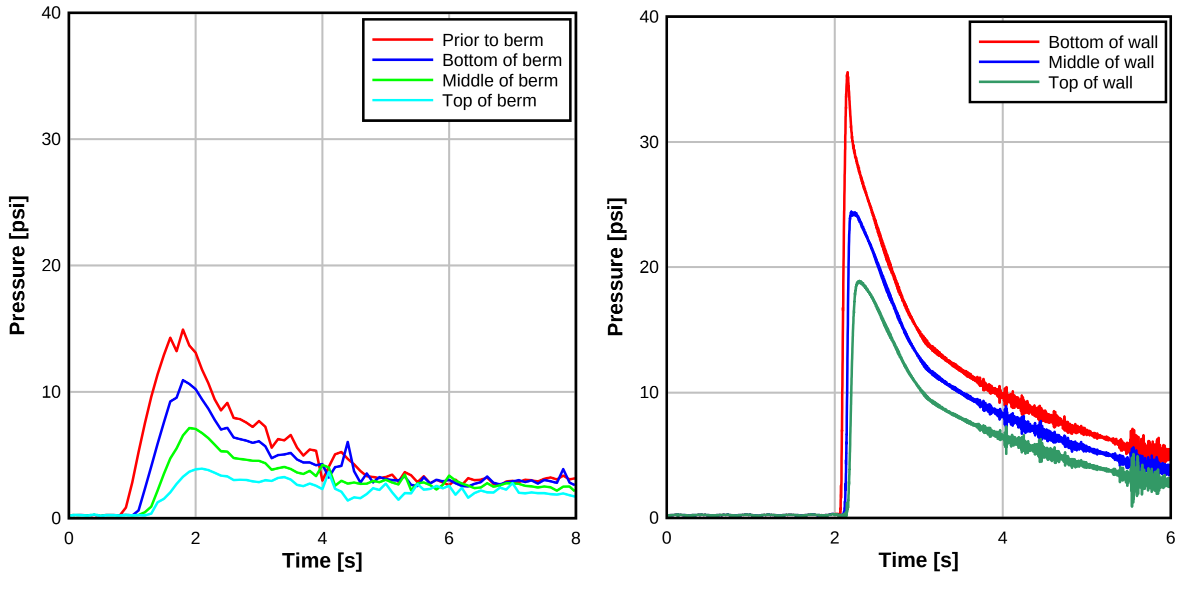


Figure 16: Calculated pressures on the berm (left) and on the retaining wall (right).

These loading histories were then used to predict the response of the berm and containment wall. In the case of the berm, the loads required to displace the soil was an order of magnitude or more than the applied loads and thus no large-scale failure was predicted. However, an SDOF model was used to predict the concrete wall response (treating the wall as a vertical cantilever and with the average load applied over the surface of the wall). The model found that it did not have sufficient resistance to survive a catastrophic release and retrofitting would be recommended.

From this study, we observe that CFD analysis can be profitably used as a means of calculating loads which then become inputs to a simplified structural response model like SDOF. Depending on the locus of complexity in a particular problem, high-fidelity models of suitable refinement can be applied as needed while simpler models may suffice for others. Tailoring the analytical approach to the problem at hand thus becomes the first task of the engineer/analyst.

Conclusions

This paper demonstrates some of the ways in which finite element analysis can be used to evaluate the response of common industrial equipment to a variety of hazard scenarios. Although derived for analysis of structures, FEA can also be a powerful tool for evaluation of such equipment as tanks, vessels, containment walls, piping, etc. Due to the complexity of the equipment, the loading, or both, simpler approaches were not sufficient to address the problems of interest. The key features of the models and analyses described above include:

- Complex interactions due to contact between tank and tank, or frame and tank, or frame and valve.
- Development of probabilistic (risk-based) failure predictions using known variabilities in material properties.
- Ability to represent the highly non-linear distribution of blast loads over a spherical or cylindrical surface.
- Ability to eliminate the need for retrofit of certain tanks, and to isolate the specific components needing retrofits when failure is predicted.
- Adaptability of CFD to provide fluid-structure interaction loads for use in simplified SDOF analysis (when the structure is sufficiently adaptable to engineering methods).

The use of high-fidelity finite element models provides increased accuracy over simpler approaches when predicting the response of complex systems. It is well suited for evaluating risk of equipment failure to business operations and pinpointing regions of failure for more targeted, and therefore less costly, retrofits and upgrades. As the available computing power and speed continues to increase and with the widespread availability of commercial FEA software like LS-DYNA, the use of FEA in safety analysis and decision making is continually becoming more widespread and cost effective.

References

J. Biggs, *Introduction to Structural Dynamics*, McGraw-Hill, Inc., New York, NY, 1964.

U.S. Department of Defense, “Structures to Resist the Effects of Accidental Explosions,” UFC 3-340-02, Change 2, September 2014.

C. Kingery and G. Bulmash, “Airblast Parameters from TNT Spherical Air Burst and Hemispherical Surface Burst,” ARBRL-TR-02555, Ballistic Research Laboratory, 1984.

Livermore Software Technology Corporation (LSTC), “LS-DYNA Theory Manual,” Livermore, CA, October 2018.