

ENHANCING OIL & GAS PLANT DESIGN WITH REALISTIC SIMULATION

VOLUME 1



Sustainable Power: Downstream Oil & Gas Companies

Content

Overview: Leveraging realistic simulation for integrated plant engineering	3
Pipe flexibility analysis and the development of PCS - Pipe Calculation System for Abaqus RMC Consultoria	4
Seismic response of floating roof storage tanks contact pressure analysis ExxonMobil Research & Engineering Co.	17
Topology optimization of a flue gas ductwork with significant pressure loss E.ON Technologies GmbH	34
Using explicit Finite Element Analysis to simulate blast loading on hazardous chemical storage tanks The Equity Engineering Group, Inc.	35
Air blast loaded safety devise dynamical response analysis Ufa State Petroleum Technological University	47
Pipeline rupture in Abaqus/Standard with ductile failure initiation Abaqus Technology Brief	55

Leveraging Realistic Simulation for Integrated Plant Engineering

ACHIEVING SAFETY, COMPLIANCE, AND PERFORMANCE WITH “NO DEFECT” PLANT ENGINEERING

“No defect” plant engineering means that your plant design and the delivered facility meet all project requirements. These large projects involve highly complex multidisciplinary engineering, procurement, and construction. They must conform to stringent regulatory, environmental, and compliance standards on both local and global levels, while meeting initial project expectations and final operational performance.

Integrated Plant Engineering **3DEXPERIENCE®** solutions enables organizations to create better project control, minimize risks, and reduce development costs through optimal execution of project development processes.



VIRTUAL TESTING OF PLANT SAFETY AND PERFORMANCE

In addition to meeting initial requirements, the plant must operate reliably and safely. Integrated Plant Engineering advanced simulation capabilities enable companies to accurately model complex real-world behaviors, such as strength and deformation in large structures, pipes, and equipment; and the impact of thermal loads, vibrations, fractures, and failures.

Volume 1 of this eBook focuses on the downstream oil & gas segment. Inside you'll find case studies, papers, and tech briefs on how leading EPC companies, owner operator (OO) engineering departments, and specialized academic institutions are leveraging realistic simulation applications to achieve expected levels of operating performance for complex facilities, while meeting high safety and environmental standards.

Realistic Simulation, as part of the Integrated Plant Engineering industry solution, helps to reduce development time and cost, improve quality and power innovation.

Piping Flexibility Analysis and the Development of PCS – Pipe Calculation System for Abaqus

Carlos Eduardo A. de C. e Sousa (RMC Consultoria, Brazil)

Abstract: The current framework for piping stress analysis is based on a simplified calculation method directly derived from experimental research performed over 60 years ago in the 1940s and 1950s. This framework was originally intended for hand calculations and, apart from minor changes and amendments, has been successfully employed by piping engineers since its development. As computational power increases and finite element analysis (FEA) becomes accessible for piping engineers, it has become clear that this framework is not well suited for complex FEA evaluation of piping. Advanced FEA procedures enable engineers to perform in depth evaluation of piping systems that are extremely difficult or even impossible to evaluate through traditional methods. Contrasting to traditional methods, a FEA simulation allow engineers to evaluate creep-fatigue interactions, advanced material models, complex loadings, complex geometries, complex support conditions, as well as the inherent evaluation of fatigue stress intensification factors. A major difficulty for performing piping FEA simulation is that currently there are no high performance FEA packages capable of efficiently generating the required analysis model for complex piping systems. This paper presents a comparison of Abaqus results and fatigue tests performed by Markl in 1952, as well as the development of PCS - Pipe Calculation System, a piping flexibility analysis application for Simulia Abaqus.

1. INTRODUCTION

Industries such as Oil & Gas, Chemical, Petrochemical, Nuclear, Naval, Mining and Pulp & Paper employ pipes as a way of moving fluids between two separate points. On petrochemical plants, it is possible to identify several acres of piping systems of varying complexity, from the simplest layouts, for long pipelines interconnecting different units, to the extremely complex layouts, for piping inside high temperature process units. Pipework represent up to 35% of the materials, 40% of the erection labor and 48% of the engineering man-hour employed on a process plant erection project (Peng & Peng, 2009). As a perspective, it is important to point out that a refinery construction costs can reach figures as high as 15 billion US dollars (Kaiser & Gary, 2013).

Piping systems are typically composed by the combination of a large number of fittings such as bends, tees, flanges, valves etc. In addition, the piping layout supports locations, and the nature and position of the installed fittings, play a major role on the mechanical response of such system.

Final piping layout and support design are highly dependent of its global mechanical evaluation and should be properly defined during design phase. When necessary, several layout iterations may be needed until a satisfactory solution is found. These iterations are time consuming and all changes must be propagated to other disciplines involved in the project (i.e.: civil works, stationary equipment, rotary equipment, instrumentation, etc.). (Basavaraju & Sun, 2004)

Even though during its operational life a piping system may experience several loading conditions, one of the most important is thermal expansion. The importance of this loading condition increases along with pipe lengths and temperatures, in many cases dictating the pipe route that must be followed.

A summary of some of the possible piping loads is presented on Figure 1.

Source: SIMULIA Community Conference, 2015

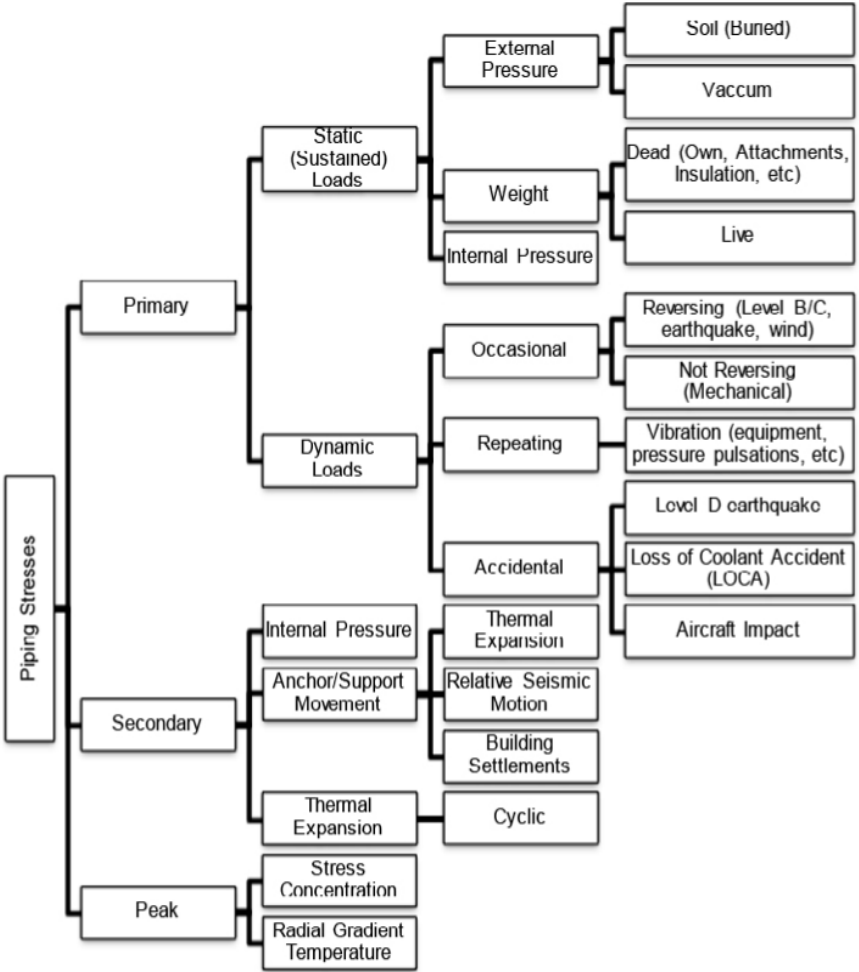


Figure 1. Piping Stresses and Loads
Adapted from (Avrithi & Agyub, 2010)

Figure 2 presents some possible failure modes that can be associated with these loading conditions based on type of loading and/or environmental condition.

Source: SIMULIA Community Conference, 2015

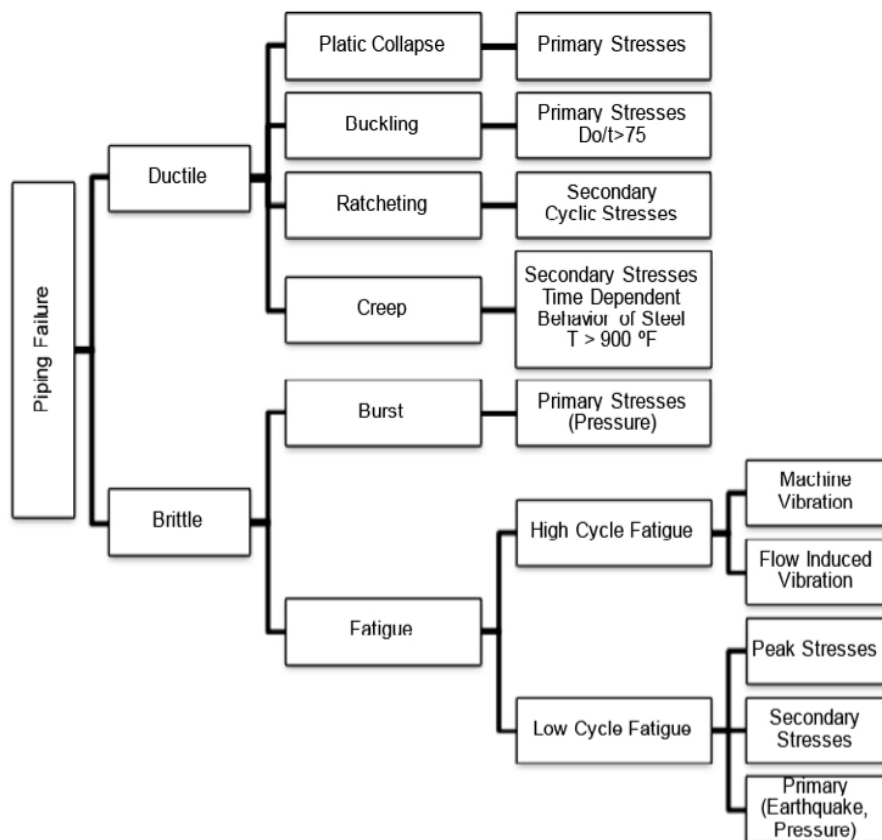


Figure 2. Failure modes for piping and their cause
Adapted from (Avrithi & Agyub, 2010)

Technological advances in chemical processes and improvements in power plants energy efficiency are often limited by the operating pressure and temperature to which their piping plants are able to safely withstand. During the design phase, increasing process temperature and pressure may be accomplished through the use of advanced materials, or the development of new ones. Usually these solutions are not cost effective or even commercially available.

Another way of improving the allowable pressure and temperature of the pipe system is through its accurate stress analysis. Improvements in stress analysis accuracy would also represent economical gain through the rational use of materials and preventing unnecessary changes in layout, as well as substantially improving the piping system operational safety and reliability.

At the present time, the vast majority of industrial pipe systems are mechanically evaluated through a simplified stress analysis procedure called "*Piping Flexibility Analysis*".

2. BACKGROUND

2.1 Piping flexibility analysis

Even though a stress analysis represents a deeper and much more comprehensive evaluation of a piping system than a piping flexibility analysis, still today many pipe engineers regard them to be equivalent (Peng & Peng, 2009).

In order to ensure safety against piping failure or its attached structures, piping flexibility analysis aims at reducing pipe stresses and loads resulting from its operation. (Markl, 1955).

A set of papers published by A. R. C. Markl (Markl, 1947, 1950, 1952, 1955) in the late 1940's and early 1950's is arguably the most significant contribution to the current development of piping flexibility methods. (Koves, 2000)

Source: SIMULIA Community Conference, 2015

A piping flexibility analysis is performed considering the pipe system as a set of straight beams. Some fittings, such as bends tees, present a reduction on their stiffness due to cross section ovalization. In order to correct its results, a set of flexibility factors is included in the analysis. This flexibility factors act as knock down factors to the stiffness whenever required.

Stress intensification factors (SIFs) need also to be included in the analysis in order to account for cyclic stresses. SIFs relate the fatigue life of a component with respect to a girth weld on a straight pipe, subjected to displacement loads.

2.2 Piping flexibility analysis inconsistencies

Even though the above framework allowed for safe design of many piping systems during the last six decades, several authors have raised questions regarding many aspects of its application (Hinnant & Paulin, 2008; Moore, Rodabaugh, Gwaltney, & Mokhtarian, 1987; Rodabaugh, 1987; Scavuzzo, 2006; E. A. Wais, Rodabaugh, & Carter, 2002; E. Wais & Rodabaugh, 1998a, 1998b)

Flexibility factors and SIFs for pipe bends and welding tees presented on (ASME-B31.3, 2012). Code standards usually provide SIFs and flexibility factors for a small amount of simple geometries. Whenever a certain geometry is not clearly permitted, the designer is required to develop this factors through testing (ASME-B31, 2008) and since the geometries provided by the Codes are very simple, this is often the case. Due to the high costs of performing such tests, many users perform FEA of the component in order to determine the required factors.

Currently no piping Code present comprehensive rules for FEA evaluation, and it is not unusual for users resort to pressure vessel codes such as (ASME-VIII-2, 2013; BS-EN-13445-3, 2012).

Mixing of piping and pressure vessel Codes is not permitted by the Codes themselves, but is an industry-accepted practice. Both types of Codes have very different design criteria and this mixing leads to an inconsistent and possibly unsafe design (Peng, 2013).

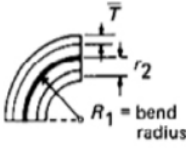
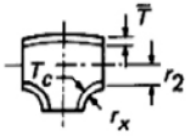
Accurate determination of the flexibility factors are of major importance. Flexibility factors are directly used on the determination of support loads, nozzle loads and, along with SIFs, in the determination of the fatigue life of the system. The piping Codes provides flexibility factors calculation formulas for pipe and mitered bends only, all other fittings have either flexibility factors of 1, or require the flexibility factor to be determined by testing.

It is clear that the Codes flexibility factor of 1 for tees and branch connections, for instance, might result on overestimation of the internal moments by a factor of 9 (Rodabaugh, 1987) or up to a factor of 20 (PRG, 2008). Inaccuracies on flexibility factors of other fittings may lead to errors of the same magnitude, while it is not possible to assure that these errors will lead to safe design.

Stress intensification factors were determined with respect to girth welds produced in the 1950's. Recent studies have shown that these Code values may need revision (Hinnant & Paulin, 2008).

These inconsistencies may lead to unnecessary re-routing of piping, incorrect estimation of loads on sensitive equipment nozzles (turbines, pressure vessels, pumps, compressors, etc.).

Table 1. Stress intensification factor (ASME-B31.3, 2012)

Description	Flexibility Factor, k	Stress Intensification Factor		Flexibility Characteristic, h	Sketch
		Out-of-Plane, i_o	In-Plane, i_o		
Welding elbow or pipe bend	$\frac{1.65}{h}$	$\frac{0.75}{h^{2/3}}$	$\frac{0.9}{h^{2/3}}$	$\frac{\bar{T}R_1}{r_2^2}$	
Welding tee in accordance with B16.9	1	$\frac{0.9}{h^{2/3}}$	$\frac{3}{4}i_o + \frac{1}{4}$		

Source: SIMULIA Community Conference, 2015

3. USING ABAQUS FOR MARKL'S FATIGUE TESTS

An alternative to avoid the inconsistencies presented on item 2.2 would be to perform the full piping stress analysis according to a modern pressure vessel Code such as (ASME-VIII-2, 2013; BS-EN-13445-3, 2012).

For a large piping system, stress analysis may become extremely time consuming and the proper use of specialized elements such as Abaqus ELBOW31 is of paramount importance. The performance of this elements have been evaluated at (Simulia, 2014).

In order to evaluate the capability of Abaqus specialized element ELBOW31 and the linear quadrilateral element S4R against the fatigue test results performed by (Markl, 1952), a set of 10 models was generated. General model dimensions are indicated in Figure 3. The fatigue evaluation of straight pipe, long radius elbow and ASME B16.9 tee were performed according to the ASME master S-N Fatigue curve (ASME-VIII-2, 2013). No safety factor on the Master Fatigue curve was considered. The cross section properties were taken according to the nominal dimensions of 4" STD pipe. Material properties according to ASTM A-106 Gr. B. For the straight pipe and long radius elbow models, both elements ELBOW 31 and S4R were evaluated. Only S4R element was evaluated for the tee model. An "opening" displacement of 25 mm on the In-Plane and Out-of-Plane directions were applied to the free edge of each evaluated system.

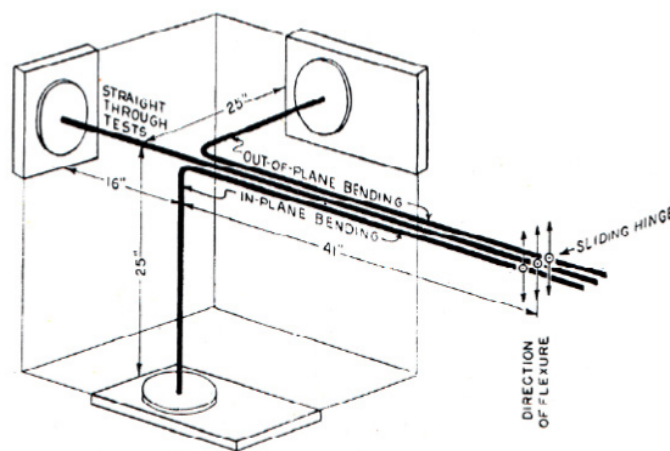


Figure 3. Markl's fatigue test general dimensions
Fig. 2 from (Markl, 1952)

3.1 Straight pipe ELBOW31 and S4R analysis

The analysis results for the straight pipe model displayed extremely good agreement to Markl's original results (Figure 5). All results from Abaqus were found to be within the test data scatter.

A sample analysis result for the S4R models is presented on Figure 4.

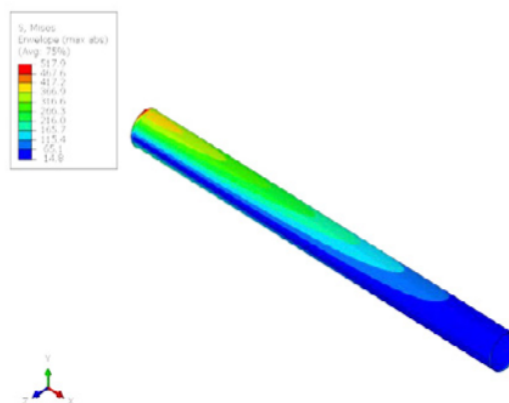


Figure 4. Sample straight pipe FEA results (S4R)

Source: SIMULIA Community Conference, 2015

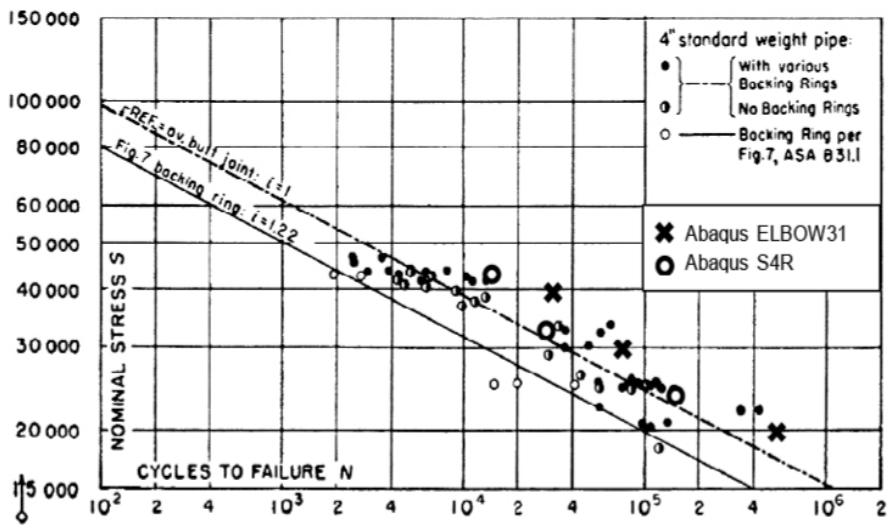


Figure 5. Fatigue results for butt-welded joints in straight pipe
Fig. 5 from (Markl, 1952)

3.2 Long radius ELBOW31 and S4R analysis

The analysis results for the long radius pipe model displayed extremely good agreement to Markl's original results (Figure 7 and Figure 8). All results from Abaqus were found to be within the test data scatter.
A sample analysis result for the S4R models is presented on Figure 6.

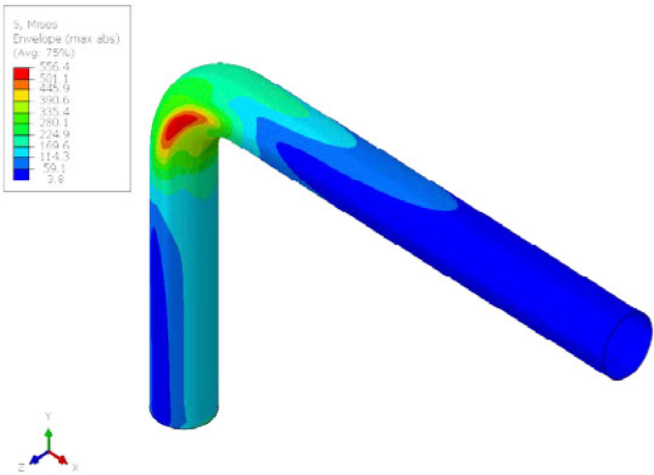


Figure 6. Sample long radius FEA results (S4R)

Source: SIMULIA Community Conference, 2015

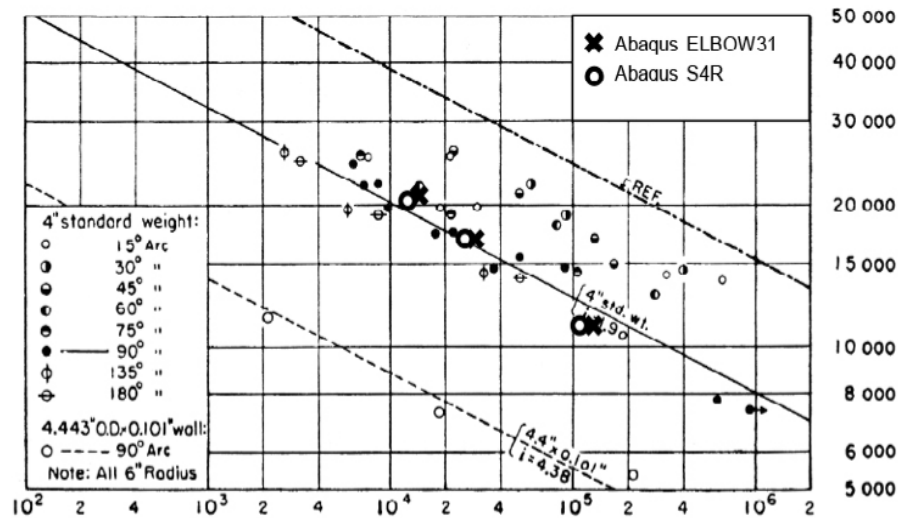


Figure 7. Fatigue results for long radius elbows, in-plane
Fig. 8 from (Markl, 1952)

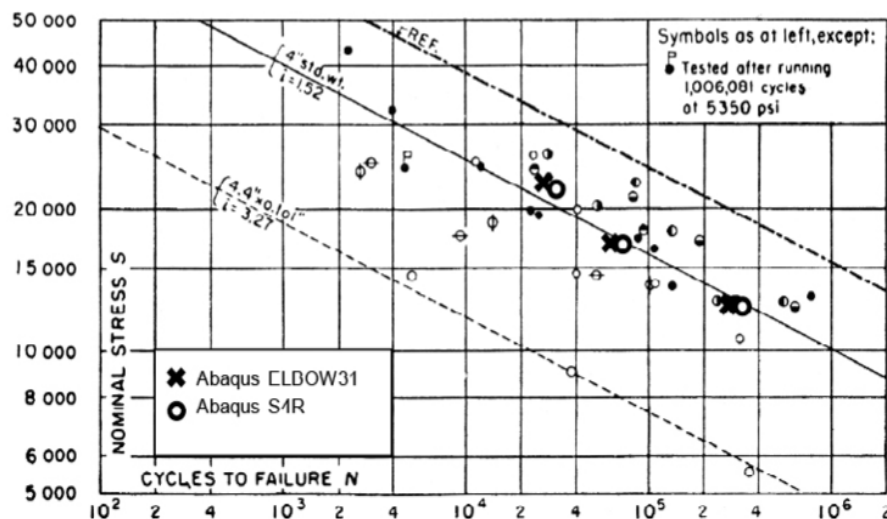


Figure 8. Fatigue results for long radius elbows, out-of-plane
Fig. 9 from (Markl, 1952)

3.3 ASME B16.9 Welding tee S4R analysis

Result analysis for the welding tee model presented good agreement to Markl's original results for the out-of-plane loading condition and Abaqus results were within the test data scatter (Figure 11). FEA results for the in-plane loading conditions were below the reference line (Figure 10). Further analysis is required, but the probable cause for the adverse result is a geometric difference between the crotch radius of Markl's welded tee and the one modeled on the present study.

A sample analysis result for the S4R models is presented on Figure 9.

Source: SIMULIA Community Conference, 2015

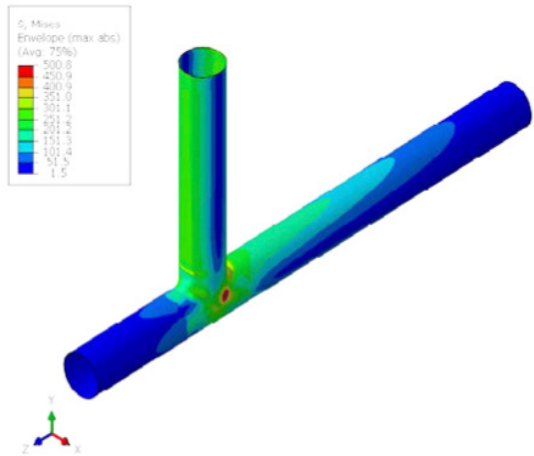


Figure 9. Sample welding tee FEA results (S4R)

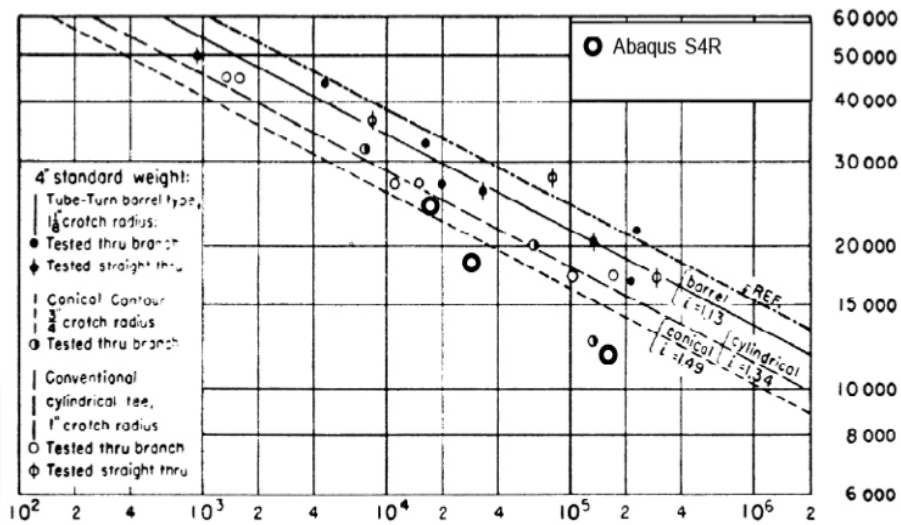


Figure 10. Fatigue results for welding tees, in-plane
Fig. 10 from (Markl, 1952)

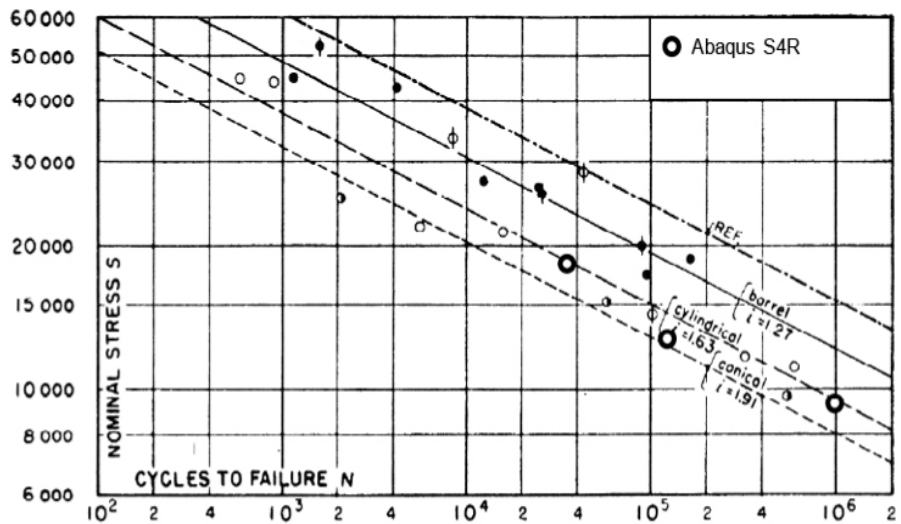


Figure 11. Fatigue results for welding tees, out-of-plane
Fig. 11 from (Markl, 1952)

Source: SIMULIA Community Conference, 2015

4. PIPE CALCULATION SYSTEM

4.1 Introduction

Finite element simulation on large industry piping systems required the development of an application for Abaqus to improve speed and ease the geometry creation process. The developed application is called Pipe Calculation System (PCS). PCS focuses in select industries such as Oil & Gas, Chemical, Petrochemical, Nuclear, Naval, Mining and Pulp & Paper.

Popular software for piping flexibility analysis currently available to end-users are Caesar II (Intergraph), Triflex (Piping Solutions) and Rohr2 (Sigma). The first two software are based on the calculation criteria originally developed by (Markl, 1955) and described on item 2, while the last one is able to perform mixed flexibility and FEA calculations. Intergraph estimated that Caesar II currently controls 75% of the piping stress analysis market share (Intergraph, 2012).

Improvements introduced by the use of PCS can be seen on the overall analysis quality, where finite element simulation provides better and more reliable results compared to regular piping flexibility. This allows the pipe system to be designed with a proper estimation of its lifespan, reducing the probability of unforeseen maintenance plant stops due to pipe failure. Additionally, PCS reduces design time with its intuitive user-interface by the use of up-to-date, consistent and reliable engineering requirements, together with easy and effective reporting with state-of-art technology.

4.2 Development

Developing such application requires merging of different expertise: a wide knowledge of the traditional framework of piping flexibility analysis; proficiency on Abaqus CAE its qualities and weaknesses regarding its use for piping flexibility analysis; knowledge on software development and Abaqus integration. With these set of skills, it is possible to define requirements and changes that must be implemented on Abaqus, in order to develop a suitable application for modern piping flexibility analysis to the end-user.

These expertise can be extended to deeper issues. Defining PCS requirements contemplates awareness of several software currently available with their strengths and weaknesses, along with how the users expect applications to react to input data and how output data should be presented.

Comprehending Abaqus can be broken down to an understanding of its internal structures, documentation, mesh generation capabilities and simulations execution. On the developer's end, integration of the application with Abaqus needs to benefit from all features already available on the software, minimizing development costs and providing foundation to PCS' results.

4.2.1 PCS Features

PCS creates an interface between the user and Abaqus in order to create the pipe system geometry, define its loads and properties, also incorporating proprietary tools previously developed by RMC to integrate PCS with different file formats broadly used. The application is also scriptable through the python interface already provided by Abaqus and a seamless user interface (UI) with Abaqus allows users to make use of the latest features developed by Simulia.

- Geometry creation

PCS allows a simplified creation of specific pipe geometries from the application's smart UI as pipes, reductions, valves and supports. Setting of material properties can be achieved by selecting it from the material database provided (). This sets the required properties such as temperature and diameter for each element modeled. This database includes properties and fittings dimensions from several piping and pressure vessels standards. The material database can also be edited, in order to modify existing values or include new ones.

- Third party file import

Previously developed pipe geometry, created on several common piping software, can be imported as seen on Figure 14. Integration to Intergraph Caesar II binary file (C2) and neutral file (CII) is available. It is also possible to import almost any 3D geometry created by all major 3D pipe design software through PCF files.

Imported data depends on the file type and amount of usable data it contains. For instance, Caesar II files allows importing geometry, temperatures, pressures and load cases, which will be automatically included in the analysis of the piping system. PCF files allow importing only basic geometry information.

Source: SIMULIA Community Conference, 2015

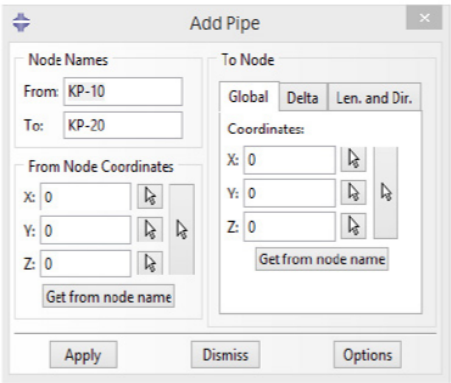


Figure 12. Pipe creation window

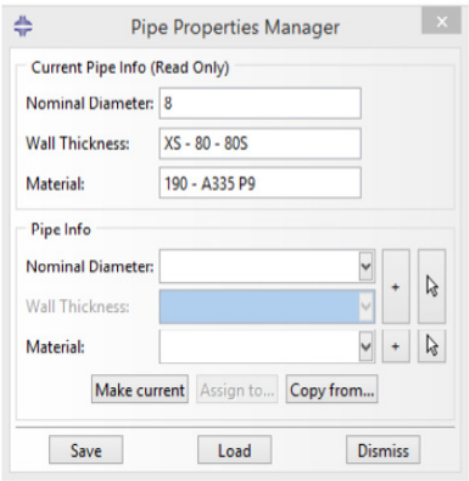


Figure 13. Pipe properties window

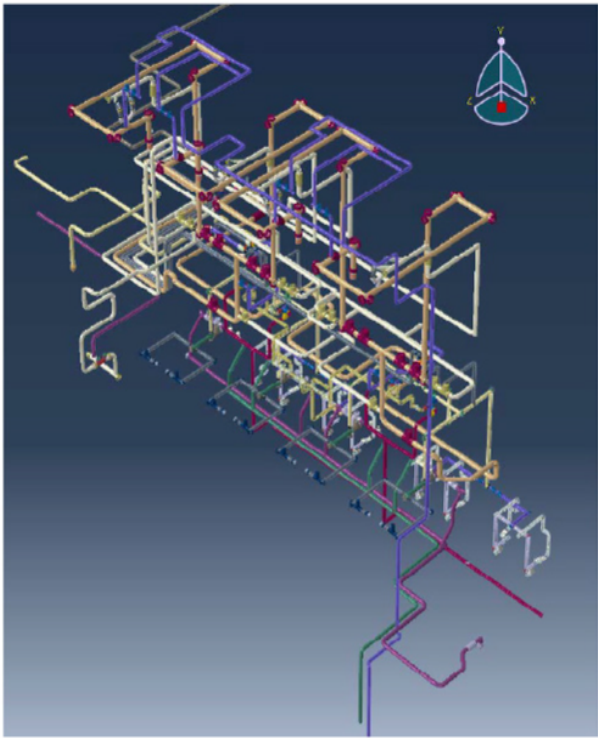


Figure 14. Caesar II Imported Model

Source: SIMULIA Community Conference, 2015

- Mesh

PCS automatically optimizes 1D meshes by generating elbow type elements, which have relatively high performance and low computational cost (Simulia, 2014). 2D meshes are also automatically generated under user request.

- Constraints

Before simulation starts all required constraints, such as elbows MPCs and support connectors, are applied to the model. Including constraints in a large piping flexibility analysis is a tedious effort that would consume considerable amount of time. This feature benefits user experience also reducing time and minimizing modeling errors.

- Reports

In order for a flexibility analysis to be checked and reviewed by a piping engineer, it has to be presented in a way that current professionals can readily evaluate, without representing a major change on the current project workflow.

PCS reports are generated in just a few user clicks. Information includes basic model data, restraint loads, displacements, stresses and allowable stresses.

4.3 Sample Results

Consider a sample piping system consisting of four parallel air-coolers. Each air-cooler has four inlet nozzles that are connected to the process unit through a 20" diameter, ¼" thick, ASTM A672 B60 pipe header. The final section of the feed pipe consists of an extremely flexible pipe arrangement in order to reduce piping loads on the equipment nozzles. The fluid is a hydrocarbon gas at 450 °C.

The PCS model consisted of 142053 linear quadrilateral elements of type S4R, geometrically non-linear and the material model was linear elastic. The evaluation of the four load steps took less than 6 minutes on a standard desktop computer (i7-4790 @ 3.6 Ghz), running Abaqus Standard on a single processor.

Figure 15 present PCS stress results for the load step WNC+P1+T1, which consists of loadings from piping weight with no contents (WNC), pipe pressure (P1), and pipe temperature (T1). These results were compared to the ASME B31.3, calculated using Coade Caesar II 5.1, a standard pipe flexibility analysis software.

The stresses computed with Caesar II converted into cycles through Markl's methodology were compared to cycles calculated according to ASME master S-N curve. The master S-N curve approach resulted on a fatigue life 20% higher than Markl's. Differences of up to 120% on nozzles reaction forces were found between the two approaches. These differences may be attributed to inaccuracies of ASME B31.3 flexibility factors that are of great importance on compact regions with several pipe fittings, such as the nozzle region of this model.

The Caesar II calculation, indicates that the piping system needs to be redesigned due to overstress, while calculation through PCS and ASME VIII div. 2 part 5 clears the system for safe operation.

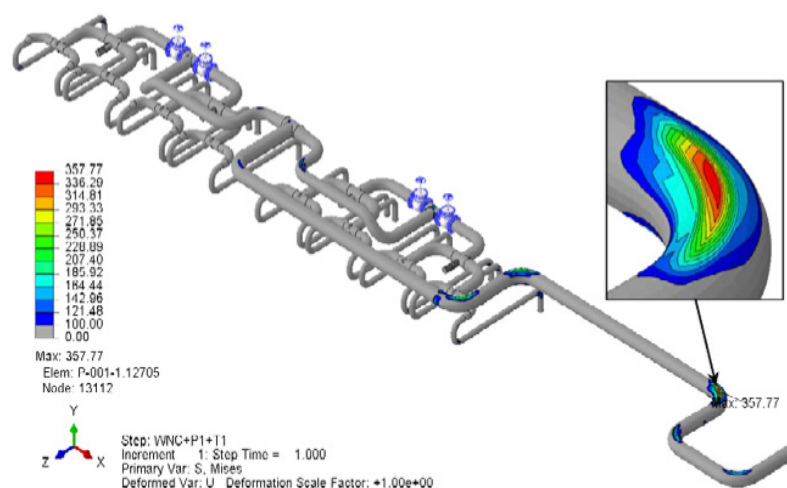


Figure 15. Sample air-cooler model

Source: SIMULIA Community Conference, 2015

5. CONCLUSIONS

On the present paper, a comparison of fatigue analysis on Simulia Abaqus and historical results from (Markl, 1952) is performed. Elements ELBOW31 and S4R were used in the FEA models while the ASME master fatigue curve (ASME-VIII-2, 2013) was used to determine the expected life of each component.

Results show that it is possible to obtain extremely satisfactory results from using Abaqus for the fatigue analyses of regular piping components.

Further analysis will be required for the welding tee under in-plane loading model, which was the only analysis case where results did not fall within Markl's data scatter. A probable cause for this result inconsistency is a geometric difference between the crotch radius of Markl's welded tee and the one modeled on the present study.

This paper also introduces PCS - Pipe Calculation System, a piping flexibility analysis application for Simulia Abaqus, a tool for fast modelling and result analysis of finite element piping systems.

A sample PCS calculation for an air-cooler pipe assembly is provided and some general results are evaluated. These results indicate a clear advantage on using advanced finite element models to evaluate common pipe flexibility analysis problems.

General use of advanced finite element software on piping stress analysis will enable users to achieve better understanding of the mechanical behavior of their piping systems, reducing calculation uncertainties, increasing fatigue life and allowing for proper evaluation of non-standard fittings. Additionally, the use of these techniques will make it possible for the piping industry to take advantage of the latest developments and trends of the simulation community.

6. REFERENCES

1. ASME-B31.1, "Power Piping", New York, N.Y., (2012).
2. ASME-B31.3, "Chemical Plant and Petroleum Refinery Piping", New York, N.Y., (2012).
3. ASME-B31.4, "Pipeline Transportation Systems for Liquids and Slurries", New York, N.Y., (2012).
4. ASME-B31.8, "Gas Transmission and Distribution Piping Systems", New York, N.Y., (2010).
5. ASME-B31J, "Standard Test Method for Determining Stress Intensification Factors (i-Factors) for Metallic Piping Components", New York, N.Y., ASME, (2008).
6. ASME-VIII-2, "ASME Boiler and Pressure Vessel Code (BPVC), Section VIII, Division 2: Alternative Rules", New York, N.Y., (2013).
7. Avrihi, K., & Ayyub, B. M., "Load and Resistance Factor Design (LRFD) of Nuclear Straight Pipes for Loads That Cause Primary Stress", *Journal of Pressure Vessel Technology*, 132, 021101, (2010), doi:10.1115/1.4000976.
8. Basavaraju, C., & Sun, W. S., "CHAPTER B4 Stress Analysis of Piping Systems", In *Piping Handbook* (pp. 107–214), (2004).
9. BS-EN-13445-3, "Unfired pressure vessels. Design", (2012).
10. BS-EN-13480-3, "Metal industrial piping - Part 3: Design and calculation", (2012).
11. Hinnant, C., & Paulin, T., "Experimental Evaluation of the Markl Fatigue Methods and ASME Piping Stress Intensification Factors", In *Volume 1: Codes and Standards* (pp. 97–113), ASME, (2008), doi:10.1115/PVP2008-61871.
12. Intergraph, "Intergraph Cadworx & Analysis Solutions Flyer", (2012), Retrieved February 10, 2015, from <http://www.intergraph.com/assets/pdf/Flyer.pdf>.
13. Kaiser, M. J., & Gary, J. H., "A Review of Refinery Markets and Cost Estimation", 597–630, (2013).
14. Koves, W. J., "Process Piping Design: A Century of Progress", *Journal of Pressure Vessel Technology*, 122(August 2000), 325, (2000), doi:10.1115/1.556199.
15. Markl, A. R. C., "Fatigue Tests of Welding Elbows and Comparable Double-Mitre Bends", *Transactions of the ASME*, 869--879, (1947).
16. Markl, A. R. C., "Fatigue Tests on Flanged Assemblies", *Transactions of the ASME*, 78--87, (1950).
17. Markl, A. R. C., "Fatigue Tests of Piping Components", *Transactions of the ASME*, 287--303, (1952).
18. Markl, A. R. C., "Piping-Flexibility Analysis", *Transactions of the ASME*, 127–149, (1955).
19. Moore, S. E., Rodabaugh, E., Gwaltney, R., & Mokhtarian, K., "Review and Evaluation of Design Analysis Methods for Calculating Flexibility of Nozzles and Branch Connections", (1987).

Source: SIMULIA Community Conference, 2015

20. Peng, L. C., "Understanding Piping Code Stress Evaluation Paradoxes And ASME B31.3 Appendix P", (2013), Retrieved December 20, 2014, from <http://www.pipestress.com/papers/piping-code-paradoxes-b31-3-p.pdf>.
21. Peng, L. C., & Peng, T. L., "Pipe stress engineering", New York, NY ASME Press, (2009).
22. PRG, "Pipe Stress Errors - When You're Off by 10 Times! Evaluating Pipe Stress Errors Due to Incorrect SIFs and Stiffnesses", (2008), Retrieved January 07, 2015, from http://www.paulin.com/WEB_FESIF.pdf.
23. Rodabaugh, E. C., "WRC-329 - Accuracy of Stress Intensification Factors for Branch Connections", Welding Research Council, (1987).
24. Scavuzzo, R. J., "Effect of Loading on Stress Intensification Factors", Journal of Pressure Vessel Technology, 128(1), 33, (2006), doi:10.1115/1.2148420.
25. Simulia, "Parametric Study of a Lner Elastic Pipeline Under In-Plane Bending", In Abaqus 6.14 - Example Problems Guide - Volume I: Static and Dynamic Analysis (pp. 1.1.3-1-1.1.3-7), (2014).
26. Wais, E. A., Rodabaugh, E. C., & Carter, R., "Investigation of Torsional Stress Intensification Factors and Stress Indices for Girth Butt Welds in Straight Pipe", In Design and Analysis of Piping, Vessels, and Components (Vol. 2002, pp. 83-89), ASME, (2002), doi:10.1115/PVP2002-1266.
27. Wais, E., & Rodabaugh, E. C., "TR-110755 - Stress Intensification Factors and Flexibility Factors for Pad-Reinforced Branch Connections", EPRI, (1998a).
28. Wais, E., & Rodabaugh, E. C., "TR-110996 - Stress Intensification Factors and Flexibility Factors for Unreinforced", EPRI, (1998b).

Source: SIMULIA Community Conference, 2015

Seismic Response of Floating Roof Storage Tanks

Contact Pressure Analysis

Philip J. Cacciatore and L. Magnus Gustafsson (ExxonMobil Research & Engineering Co.)
Alexander Kozak (SC Solutions Inc.)

Abstract: Seismic response of liquid storage tank floating roofs involve phenomena that require dynamic nonlinear geometric and material behavior as well as surface to surface contact. Good engineering practice requires a practical analytical approach that captures the essential ingredients of structural behavior under earthquake excitation by making reasonable, conservative, and manageable approximations to the actual conditions. This paper discusses an approach used in ABAQUS to calculate the stresses and deformations of a liquid storage tank floating roof under seismic loading. It represents a novel application of contact theory to achieve a solution to this problem. The method is validated by a fully coupled fluid-structure interaction (FSI) finite element analysis using actual earthquake ground accelerations. The method is supported by both the American Petroleum Institute (API) and the Petroleum Association of Japan (PAJ).

INTRODUCTION

The large capacity liquid storage tank is an important facility for the storage of pre-production raw materials and partially or completely processed products for the petrochemical industry. These structures are primarily cylindrical in shape, with a very flexible bottom plate and either a fixed, floating, or combination of fixed and floating roofs. The underlying foundation for the tank can be asphalt, concrete, pre-designed or natural soil support, or crushed stone. These tanks are used to store many different types of hazardous and volatile liquids such as crude oil, naphtha, and gasoline.

The storage tank is a relatively simple welded or riveted structure but it responds to most loadings in a highly nonlinear manner. Beyond the basic hydrostatic loading of the contents, it can be subjected to large deflections and buckling from foundation settlement, wind, rain, snow, and seismic loading. These loads can induce localized plastic deformations in the tank bottom plate, shell walls, and roof.

The focus of this paper is the seismic liquid sloshing response of floating roof storage tanks (Figure 1). Seismic loading will induce both impulsive (fluid mass) and convective (liquid sloshing) loads on the tank walls. Simplified seismic design attempts to minimize the effects of these loadings which generate shearing and overturning forces on the tank. Primary mitigation methods of seismic effects rely on anchoring systems based on a seismic design spectrum. Earthquake loading is random and can be so powerful in magnitude that anchoring systems can and often fail. Inadequate design could result in loss of containment integrity or the more unusual "walking away" off the foundation and a phenomena known as "elephant foot buckling" (Cacciatore, 2004). Floating roof tanks are particularly vulnerable to seismic events since the displacement of the roof is driven by the induced sloshing waves generated on the liquid surface (Kikuchi, 2005). If key floating roof components fail and roof buoyancy is lost, the roof will sink, exposing the environment to the hazardous material and creating the potential for a fire and/or explosions. If the seismic effects can not be mitigated, liquid levels must be lowered to reduce the seismic impulsive loadings and the sloshing wave effects on the roof. This can result in a significant reduction in capacity with a corresponding economic impact.

CONTACT PRESSURE APPROACH (CPA)

The analysis of a single floating roof tank under seismic loading subject to all the nonlinear behavior described above is a formidable task. Engineering analysis requires a practical analytical approach that captures the effects of the seismic loading by making reasonable, conservative and manageable approximations to the actual conditions. The goal is simply to obtain results that can guide design and design modifications and maintain an adequate safety level. ABAQUS provides the necessary tools to implement an approach based on standard finite element modeling and surface contact algorithms.

The Contact Pressure Approach (Cacciatore, 2009) described in this paper uses the Velocity Potential theory as a starting point for the analysis (ASCE Technical Council, 1984). Velocity Potential is a linear theory that is limited in scope. The Japanese Fire Disaster Management Agency, due to their strong interest in the damage from seismic response, has completed numerous studies and experiments on floating roof tanks. These have resulted in the addition of a nonlinear component to the linear Velocity Potential Theory (Japanese Fire Disaster Management Agency, March 2006) (In Japanese). The modified equations formed the basis for additions to the Japanese Fire Service Law (FSL) (Yamauchi, 2006), and they are used in a simplified spreadsheet based analysis.

Source: SIMULIA Community Conference, 2015

Based on the frequency content of most earthquakes and typical tank-liquid systems, a modal synthesis of the response of the first and second modes of sloshing will capture the main characteristics of the overall response. The modified Velocity Potential equations for 1st ($\eta_1(r, \theta)$) and 2nd mode ($\eta_2(r, \theta)$) sloshing are shown in equation form below and graphically in Figures 1 and 2. Note that the rigid body motion of the mode has been removed ($-r/R$) and the displacement shapes are just deviations from the rigid body plane.

$$\eta_1(r, \theta) = \beta * (\eta_{\max}^{(1)} \cos \theta + 0.8 * R * (\frac{\eta_{\max}^{(1)}}{R})^2 \cos 2\theta) (\frac{J_1(\lambda_1 \frac{r}{R})}{J_1(\lambda_1)} - \frac{r}{R}) \dots (1)$$

$$\eta_2(r, \theta) = \alpha * \eta_{\max}^{(2)} \left(\frac{J_1(\lambda_2 \frac{r}{R})}{J_1(\lambda_2)} - \frac{r}{R} \right) * \cos \theta \dots (2)$$

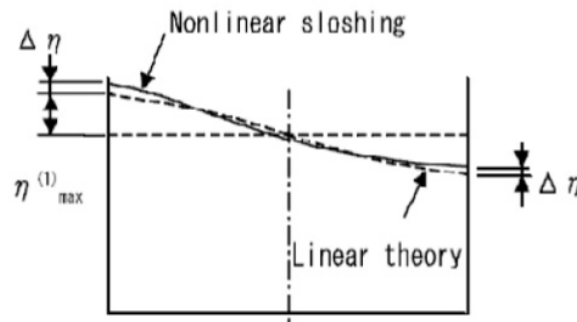


Figure 1. Mode 1 linear & nonlinear sloshing components.

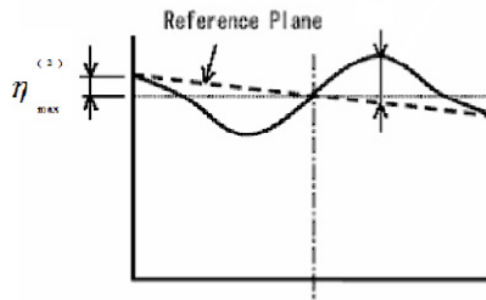


Figure 2. Mode 2 linear sloshing component.

The basic equations have been further modified by ExxonMobil with the addition of two factors, α and β , which represent the attenuation fraction for each mode. The attenuation factors modify the response to account for the attenuation of the free surface sloshing wave by the presence of the floating roof on the liquid surface. The second term (shown in red) in Equation 1 is $\Delta\eta$ and the coordinate system is polar (r, θ). The function J_1 is the standard 1st order Bessel function, and the λ_i are the roots of the derivative of the 1st order Bessel functions. The key parameters in the Mode 1 and 2 sloshing wave equations are $\eta_{\max}^{(1)}$ and $\eta_{\max}^{(2)}$. These values are the maximum sloshing height at the tank wall for each mode. These parameters, which are based on the Velocity Potential Theory, are provided below:

Source: SIMULIA Community Conference, 2015

$$\eta_{\max}^{(1)} = \frac{D}{2g} * \frac{2}{\lambda_1^2 - 1} * \left(\frac{2\pi}{T_1} \right) * S_v^{(1)} \quad \dots (3)$$

$$\eta_{\max}^{(2)} = \frac{D}{2g} * \frac{2}{\lambda_2^2 - 1} * \left(\frac{2\pi}{T_2} \right) * S_v^{(2)}$$

$$\omega_j^2 = \frac{\lambda_j g}{R} \tanh\left(\lambda_j \frac{H}{R}\right)$$

$$T_j = \frac{2\pi}{\omega_j}$$

λ_j = j th root of the derivative of the 1st order Bessel Function

R = radius of the tank

H = height of liquid in the tank

where T_1 , T_2 , $S_v^{(1)}$, and $S_v^{(2)}$ are respectively the periods of vibration for the 1st and 2nd mode tank-liquid system and the pseudo velocity spectrum values corresponding to a frequency sweep (Duhamel Integral evaluation) of the ground acceleration data. D is the tank diameter and g is the acceleration of gravity. Note the dependence of the period on the liquid height and the corresponding dependence of sloshing wave height on period.

Figure 3 contains a comparison between Fluid Structure Interaction (FSI) analysis and Velocity Potential Theory for Mode 1. The FSI analysis is performed with ADINA and employs fluid potential based elements. The FSI results are inherently linear in approximation (frequency and mode shape calculation are inherently a linear perturbation process). The FSI results show good agreement with Velocity Potential Theory linear results.

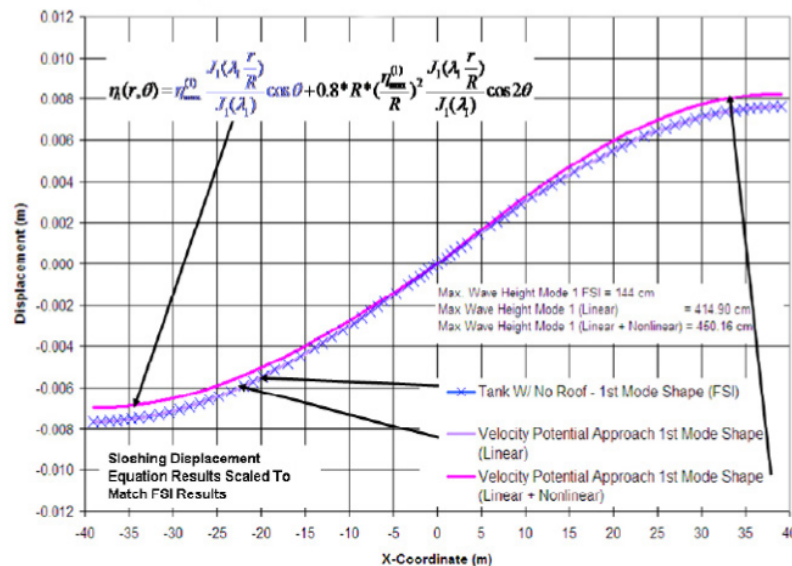


Figure 3. First mode comparison velocity potential & fluid-structure interaction models.

Source: SIMULIA Community Conference, 2015

Figure 4 shows a similar comparison for Mode 2. Both of these calculations are validation of the FSI model.

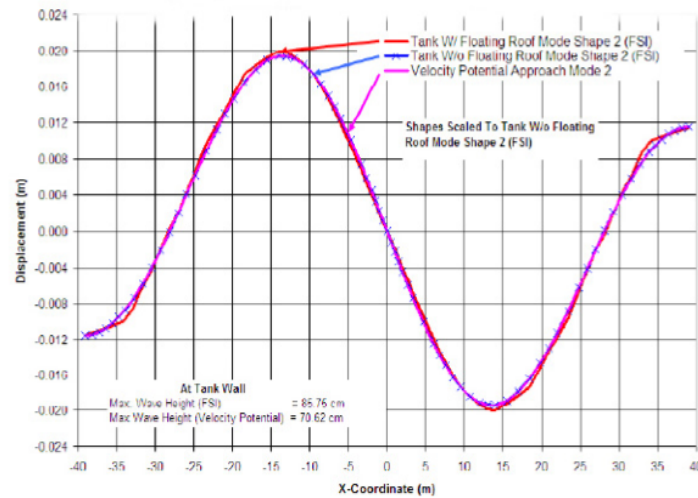


Figure 4. Mode 2 comparison of velocity potential & fluid-structure interaction models

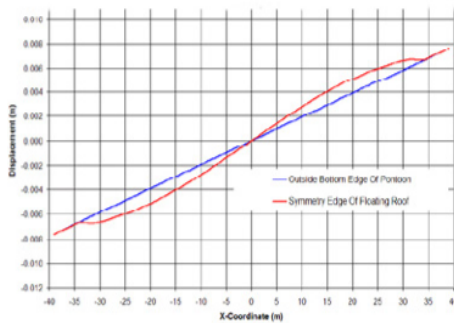


Figure 5. Mode 1 FSI w/ floating roof.

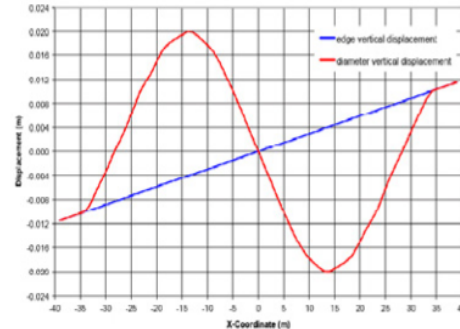


Figure 6. Mode 2 FSI w/ floating roof.

Figures 5 and 6 contain Modes 1 and 2 with the floating roof included in the FSI model. The general form of Modes 1 and 2 are apparent however, the presence of the pontoon introduces a discontinuity in the smooth Velocity Potential mode shapes. The floating roof deck (center plate) is flexible enough to follow the sloshing wave. The pontoon structure is however fairly rigid and appears to simply undergo rigid body motion. In the θ direction, the pontoon has a finite bending rigidity and is immersed in the sloshing liquid. There is a variation in the pressure on the pontoon which is proportional to the difference in the immersion depth and the $\Delta\eta$ nonlinear component of Mode 1. The pontoon displacement from the rigid plane can be developed by examining the deformation of ring beam subject to a uniform load (Japanese Fire Disaster Management Agency, March 2006).

$$\omega_j^2 = \frac{\lambda_j g}{R} \tanh\left(\lambda_j \frac{H}{R}\right)$$

$$T_j = \frac{2\pi}{\omega_j}$$

λ_j = j th root of the derivative of the 1st order Bessel Function

R = radius of the tank

H = height of liquid in the tank

The larger the bending rigidity ($\frac{8EI_\theta}{R^4}$), the smaller ϕ will be and the displacement approaches $\Delta\eta$ (nonlinear component of Mode 1 sloshing wave). Eliminating $\Delta\eta$ from the sloshing wave leaves the pontoon with rigid body motion only.

Source: SIMULIA Community Conference, 2015

The key ingredients in the sloshing wave equations are sloshing wave heights at the tank wall, i.e. $\eta_{\max}^{(1)}, \eta_{\max}^{(2)}$. These quantities must be closely monitored in the analysis since they can exceed the distance between the static liquid surface and the top of the tank wall (freeboard distance). This is an indication of an overflow/spillage condition which has implications for the sloshing wave effects on the floating roof. It is conservative to ignore the overflow condition, but it can easily be accounted for in the analysis by assuming the actual wave height at the wall can only reach a maximum value equal to the freeboard. Using this assumption, the component modes can be adjusted to stay within that limitation.

The contact pressure concept is used in two separate phases of the analysis. The first phase consists of determining the attenuation factors for the individual sloshing modes. The second phase is the process of modal synthesis (usually square root sum of the squares) of the two main sloshing modes and applying the modal deformations to the floating roof. The following summarizes the Contact Pressure Approach for calculating attenuation factors:

1. The attenuation factor for each mode depends upon:
 - a. Pontoon Geometry
 - b. Deck Stiffness
 - c. Velocity Spectrum Values - $S_v^{(i)}$ which depend on the natural period of vibration for the tank-liquid modes (T_i) which, in turn, depends on among other things the tank diameter and liquid height.
2. Using the Velocity Potential Theory for sloshing wave displacements of the floating roof precludes buckling of the roof components in contact with the liquid. Imposing displacements on the roof is not realistic. A displacement controlled application of the sloshing wave needs to be replaced by a "sloshing pressure" which will give the roof the opportunity to respond to the sloshing wave in a physically realistic manner.

CPA Process - Determination of α and β Modes 1 & 2 Attenuation Factors

- a. A membrane is placed under the floating roof (Figure 7). Structural details of a typical roof are shown in Figure 8.
- b. The floating roof is allowed to settle onto the membrane.
- c. As the roof passes through the membrane, the membrane develops a contact pressure proportional to the distance the roof sinks past the membrane. Contact Pressure = $\rho g \times \text{distance}$.
- d. The contact pressure equalizes to the weight of the roof. This is the buoyant state of the floating roof. Figures 9 & 10 show the contact "buoyancy" pressure and displacement from dead weight loading.
- e. The membrane is now displaced incrementally into one of the sloshing "mode shapes". At each increment of displacement, the contact pressure between the membrane and roof is calculated. The membrane can be displaced into either the Mode 1 or Mode 2 sloshing wave. Figure 11 shows the contact pressure at a fraction of the full Mode 1 sloshing wave.
- f. The Mode 1 shape has a high and low points at $\theta=0^\circ$ and 180° on the symmetry plane, The "red" areas of Figure 11 indicate areas of high pressure and the white areas represent no pressure or loss of contact.
- g. The blue areas indicate areas of "low" pressure and these will occur in two possible locations:
 - ▲ Where the deck has buckled and possibly separated from the liquid surface.
 - ▲ Where the perimeter of the pontoon may be "lifting out" of the liquid.
- h. The CPA method can predict the "potential" for pontoon-liquid separation. This opens the possibility of liquid escaping onto the deck or spilling over the tank top and down the outside tank walls. However, separation can occur without spillage. Mode 1 produces the highest wave height at the tank wall. Figure 12 show the Mode 1 formation and shape minus rigid body motion.
- i. The white area along the edge of the pontoon in Figure 11 indicates a zero contact pressure (loss of contact) condition for this point in time. If we go backwards in time (reducing the fraction of the wave) to that point where the loss of contact develops, we can use the wave fraction to set the attenuation factor for Mode 1. Keep in mind the true behavior is dynamic and as soon as the loss of contact occurs along the periphery in Mode 1, fluid will rush in to equalize the pressure under the pontoon.

Source: SIMULIA Community Conference, 2015

- j. A zero pressure on the bottom of the floating roof is not physically realistic. The higher pressure areas of the wave will collapse and push liquid into the lower pressure regions. This prevents the wave from reaching its maximum wave height resulting in "attenuation" of the modal response.
- k. For Mode 2, the membrane is displaced incrementally into the Mode 2 displacement pattern and the contact pressure monitored on the bottom of the roof.
- l. Figure 13 contains the Mode 2 sloshing wave position following the initial buoyancy establishment of weight equilibrium.
- m. Certain zones under the roof continue to develop pressure according to the $p \propto x$ distance relation (pressure-over-closure relationship). This is the wave crest.
- n. Other zones of the roof will experience a decrease in pressure since the wave is moving away from the roof. This is the wave trough.
- o. The roof will follow the wave crest. However, it can only follow the wave trough as far as physically possible (via gravity). At that point the roof will become a catenary over the trough and the contact pressure will go to zero.
- p. The fraction of the imposed Mode 2 sloshing wave on the membrane at the point of initial zero contact pressure will set the attenuation factor.
- q. Once again, fluid pressures in the crest area will drive fluid into the trough in attempt to equalize underneath the roof.
- r. It is important to keep in mind that these are dynamic events that are treated statically. If the event is very slow, the collapse will begin earlier than if the wave formation was a high speed event.
- s. Note that the bottom of the floating roof (including the bottom of the pontoon) is allowed to buckle. This would not be possible if the sloshing wave's displacements were imposed directly on the bottom of the roof.
- t. If the roof is able to achieve the full sloshing wave (Mode 1 or 2) without developing zero contact pressure there is no attenuation. If the nonlinear incrementation of the sloshing wave fails before there is loss of contact, then the roof has buckled and failed.

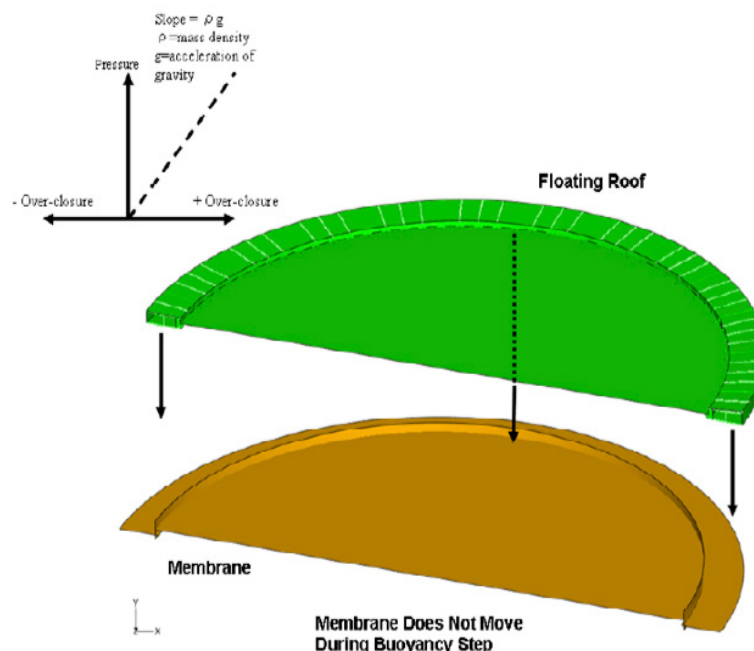


Figure 7. Dead weight loading of floating roof on to membrane.

Source: SIMULIA Community Conference, 2015

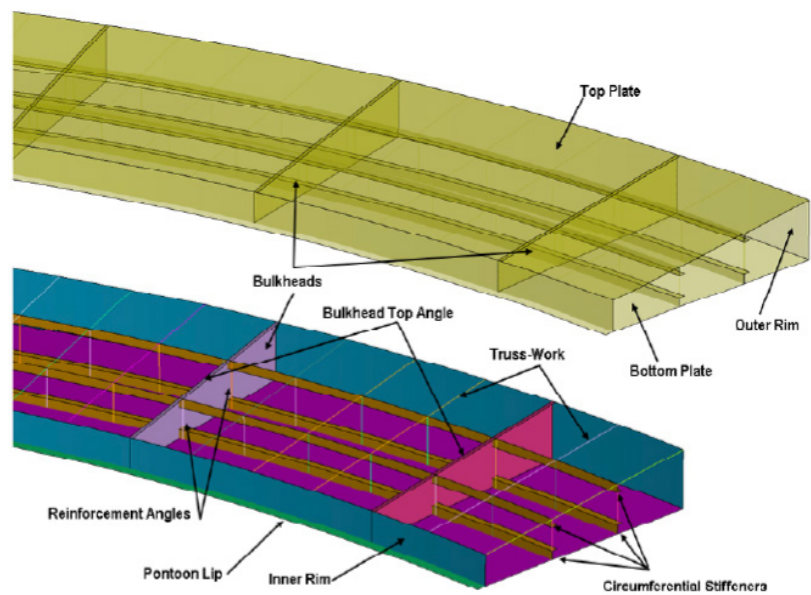


Figure 8. Details of typical pontoon structure.

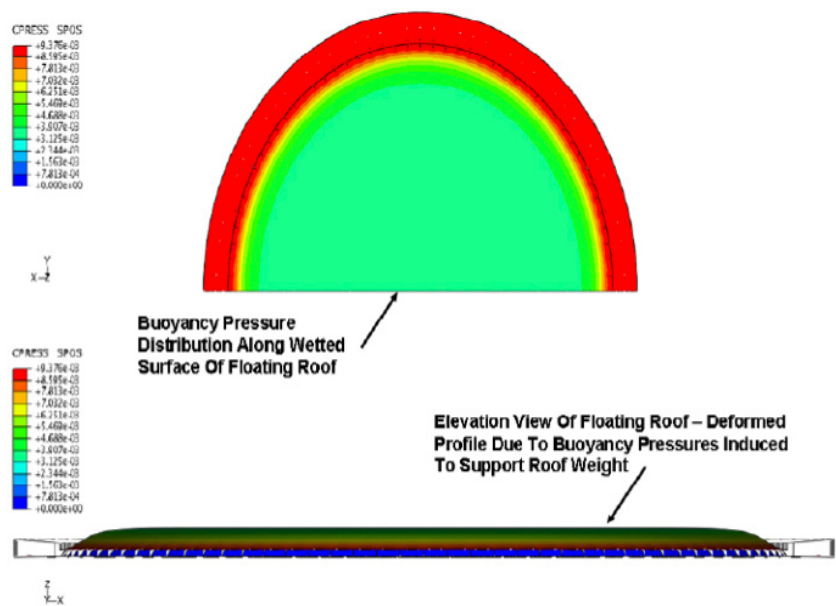


Figure 9. Dead weight pressure distributions on floating roof.

Source: SIMULIA Community Conference, 2015

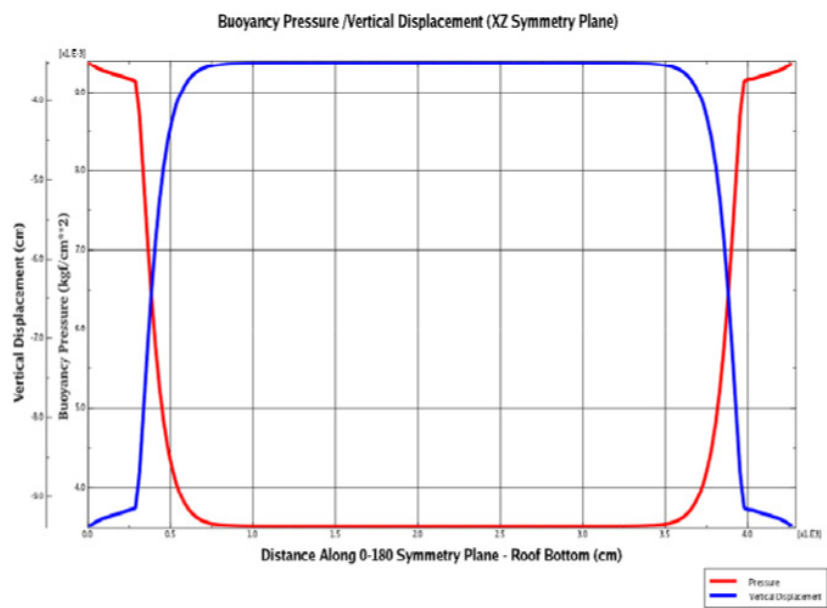


Figure 10. Buoyancy pressure and displacement along symmetry plane of roof.

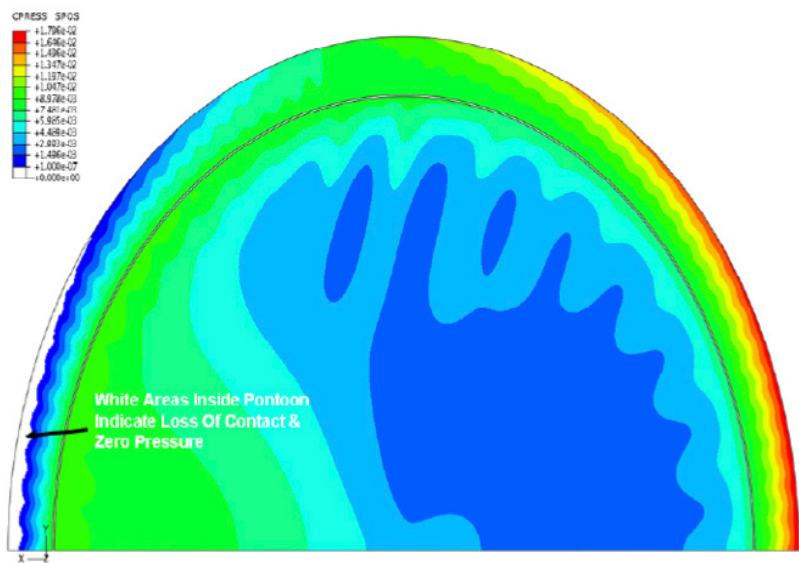


Figure 11. Contact pressure on floating roof at 31.1% of fully developed mode 1 sloshing.

Source: SIMULIA Community Conference, 2015

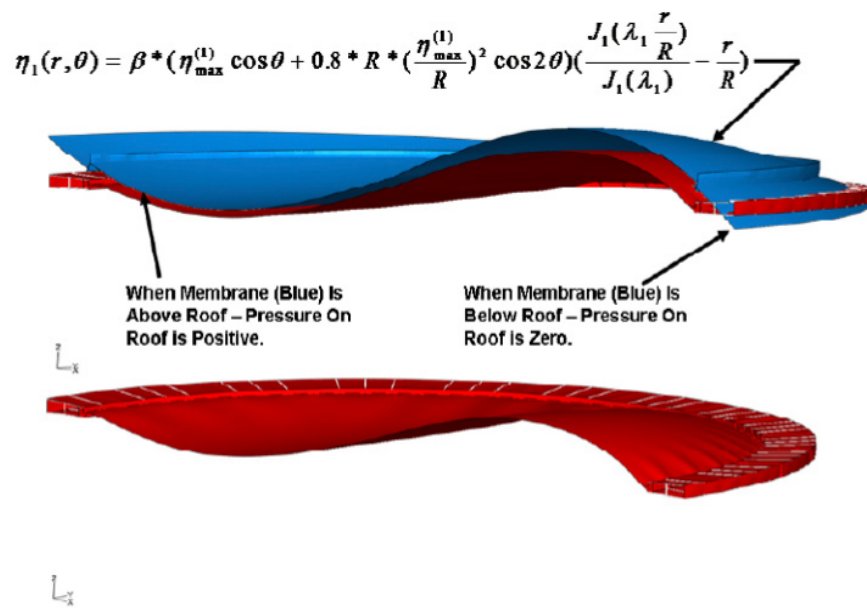


Figure 12. Floating roof w/wo membrane - Mode 1 attenuation factor calculation.

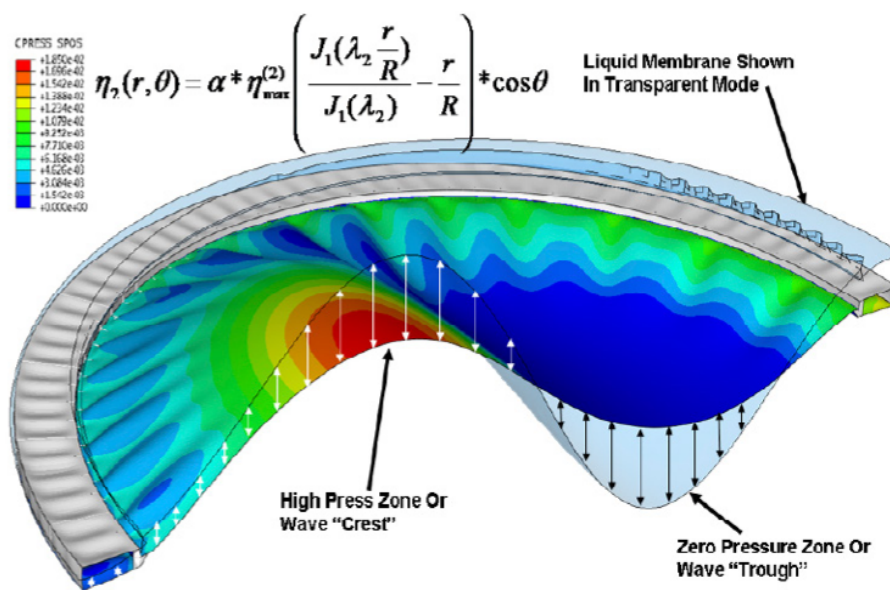


Figure 13. Mode 2 shape & attenuation factor calculation.

3. Calculation of Displacements, Forces, Moments and Stresses

- Once the attenuation factors have been calculated for Modes 1 and 2, there are a number of options for determining the displacements, forces, moments and stresses.
- The most accurate and technically defensible is to combine the attenuated Mode 1 and Mode 2 displacement shape into a single synthesized deformation pattern.

Source: SIMULIA Community Conference, 2015

- c. This is accomplished by calculating the magnitude of displacement for each mode at each point in the pontoon. A square root of the sum of the squares of the magnitude is calculated and then a cosine variation is restored to the pattern.
- d. The synthesized sloshing deformation pattern is then applied to the deck portion of the floating roof first. This displacement pattern reflects the presence of the pontoon on the floating roof. This step calculates the critical radial contraction displacements of the deck on the pontoon.
- e. In the next step, refine the pontoon element mesh for accurate force and stress calculation. Apply the radial contraction from (d), dead weight and buoyancy loads to the pontoon while subjecting the pontoon to the deformations describe in Equation 4.
- f. All calculations are based on large geometric deformations.
- g. Calculate the resulting deflections, forces and stresses in the pontoon under large geometric deformations.
- h. Evaluate critical weld strength in separate detailed analyses.

ANALYSIS OF 100 KKL TANK

The concepts above are applied to a 100 KKL located on the Japanese Island of Hokkaido. In 2003, a magnitude 8.0 earthquake (Tokachi-Oki) occurred offshore of Hokkaido in the Kuril Trench which is part of the Pacific Ring of Fire. The earthquake caused extensive damage to numerous storage tanks at a Japanese Refinery on the island. The 100 KKL tank was one of the largest damaged tanks. Critical parameters of the tank and the results of simplified FSL spreadsheet calculations for stress are shown below in Figure 14.

The spreadsheet calculations are based on methods incorporated in the Japanese Fire Service Law (FSL). The law set the minimum requirements to determine if a floating roof required modifications to sustain earthquake loads. The modifications are not defined and the methods could be manipulated to produce the desired result without improving the integrity under earthquake loading.

FSI MODEL

A finite element model for the tank wall, tank bottom, the contained fluid and the floating roof is used in a dynamic, direct integration fluid-structure interaction (FSI) analysis in ADINA (Kozak, 2008).

1. Assume quake response is symmetrical with respect for the earthquake horizontal direction
2. Assume tank wall and bottom are rigid. These components are primarily affected by the impulsive loads of the contained fluid. Impulsive loads occur in the 2 Hz range whereas the convective 2nd Mode sloshing is in the 0.2 Hz range.

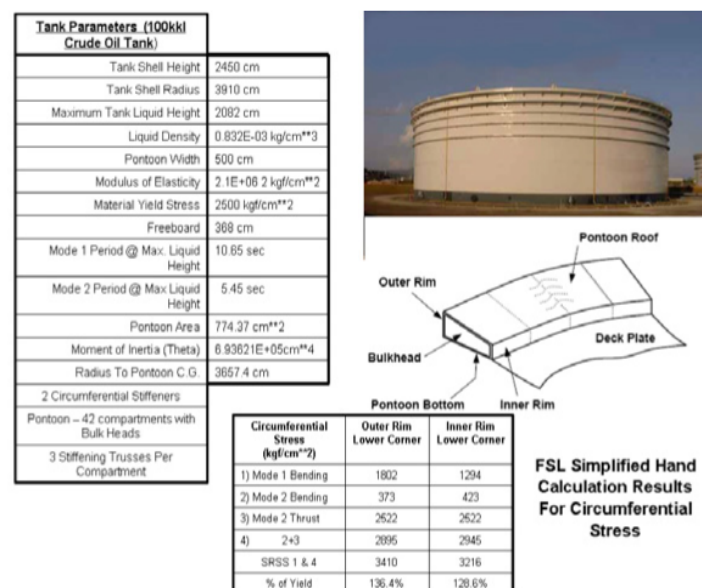


Figure 14. Example floating roof tank analysis.

Source: SIMULIA Community Conference, 2015

3. Tokachi-Oki earthquake of 2003 is used in the analysis. Only the horizontal (East-West) component of the ground acceleration is used. The acceleration is applied to the tank bottom.
4. Earthquake spectra applied in 3 analyses:
 - a. FSI model constrained to respond in the 1st Mode only
 - b. FSI model constrained to respond in the 2nd Mode only
 - c. Unconstrained FSI model constrained to respond in combination of 1st and 2nd modes
5. FSI model verified with dead weight and frequency analyses.
6. Stiff (compression only) interaction between the floating roof and tank shell wall.

Table 1. FSI vs FSL frequency, mode shape, and participation factor calculation.

Tank W/o Roof Frequency Analysis

Mode	f (Hz)	Period (sec)	Modal Participation Factors	
			Y	Z
1	0.0939	10.652	64.3%	0.0%
2	0.1340	7.463	0.0%	3.85%
3	0.1534	6.519	0.0%	5.23%
4	0.1615	6.192	0.0%	0.0%
5	0.1632	5.459	0.0%	0.04%
6	0.1834	5.453	2.6%	0.0%

Tank W/ Roof Frequency Analysis

Mode	f (Hz)	Period (sec)	Modal Participation Factors	
			Y	Z
1	0.0937	10.676	64.2%	0.0%
2	0.1384	7.225	0.0%	0.0%
3	0.1535	6.515	0.0%	0.0001%
4	0.1833	5.456	2.5%	0.0%

FSL Equations

Mode 1 = 10.65 sec

Mode 2 = 5.45 sec

Table 1 contains the modal frequency response and participation factors for a floating roof storage tank with and without the roof. The presence of the floating roof has a minor impact on both quantities. The earthquake ground acceleration record used in the time history FSI analysis has been used to generate a pseudo-velocity spectrum (Figure 15). The velocity potential equations require a modal velocity factor for each mode. Figure 15 indicates that for periods of 5.46 and 10.68 seconds the pseudo-velocity values are 237.6 and 70.7 cm/sec respectively.

Source: SIMULIA Community Conference, 2015

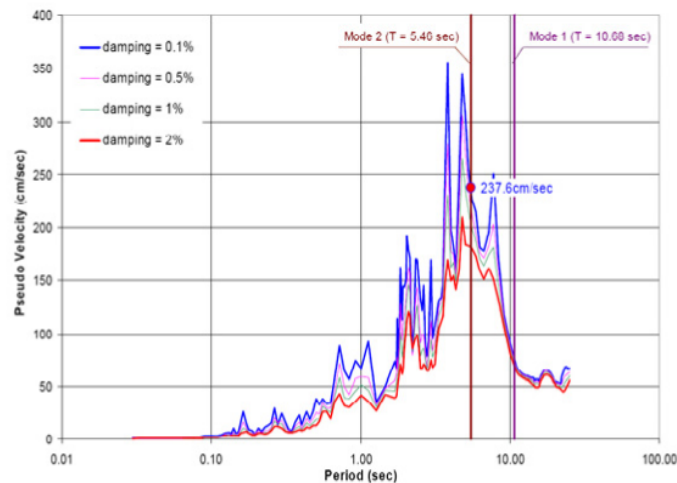


Figure 15. Pseudo velocity spectrum for 2003 Tokachi-Oki earthquake.

The Mode 1 surface oscillation retrieved from the free surface and floating roof analyses are compared in Figure 16. These diagrams match very well which indicates that there is no Mode 1 attenuation due to the floating roof for this tank under Tokachi-Oki ground accelerations.

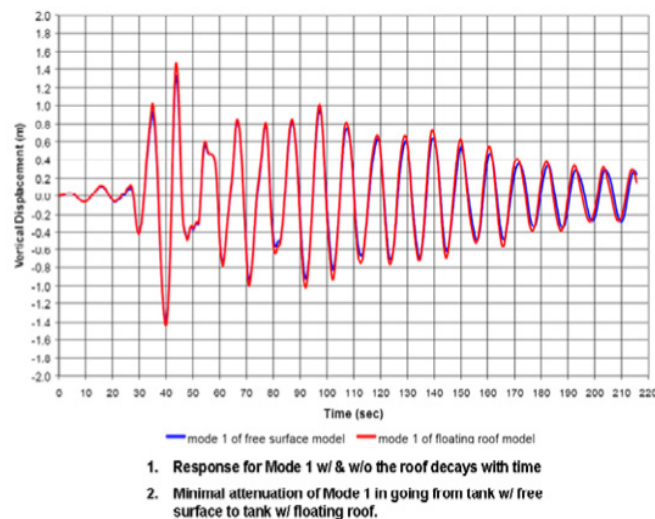


Figure 16. FSI Mode 1 constrained fluid under 2003 Tokachi-Oki ground accelerations.

The FSI results are in good agreement with the CPA. The Contact Pressure Approach predicts a Mode 1 $\beta=1.0$, i.e. no attenuation. This agrees with the FSI solution. Comparison of the Mode 2 response in Figure 17 indicates a strong attenuation of the sloshing wave of the free surface due to the floating roof.

Source: SIMULIA Community Conference, 2015

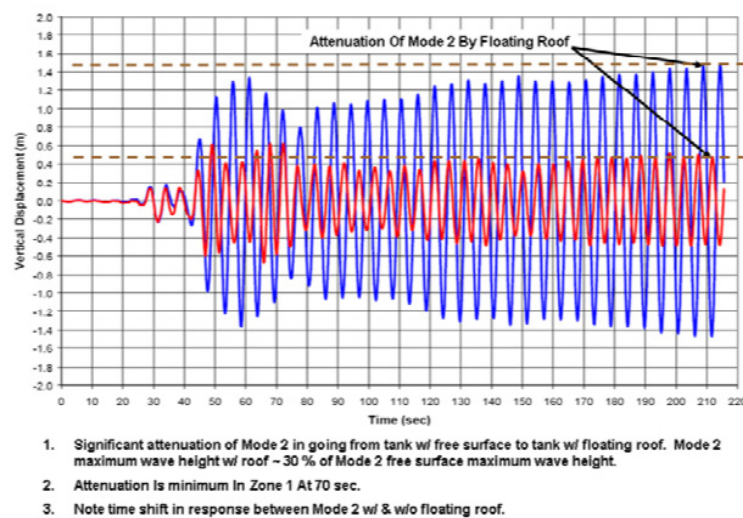


Figure 17. FSI Mode 2 constrained fluid under 2003 Tokachi-Oki ground accelerations.

The Contact Pressure Approach predicts a $\alpha=0.2623$ based on a velocity spectrum value of 210.8 cm/sec, the maximum allowed in the velocity spectrum specified in the Japanese FSL. The FSI α ranges from 0.275 to 0.32 but corresponds to the pseudo-velocity value of 237.6 cm/sec.

Figure 18 contains the time history response of the total maximum wave height over the entire surface of the floating roof (blue line) and maximum wave heights at the tank wall. Note that total maximum wave height on the roof occurs at different points on the roof in the time domain of the response. The red line mostly overlaps the blue one (total maximum at the wall) which indicates domination of the Mode 1 sloshing. This is confirmed in Figure 19 which demonstrates the total maximum wave height (blue line) and the Mode 1 and Mode 2 contributions (red and green lines respectively)

RESULTS

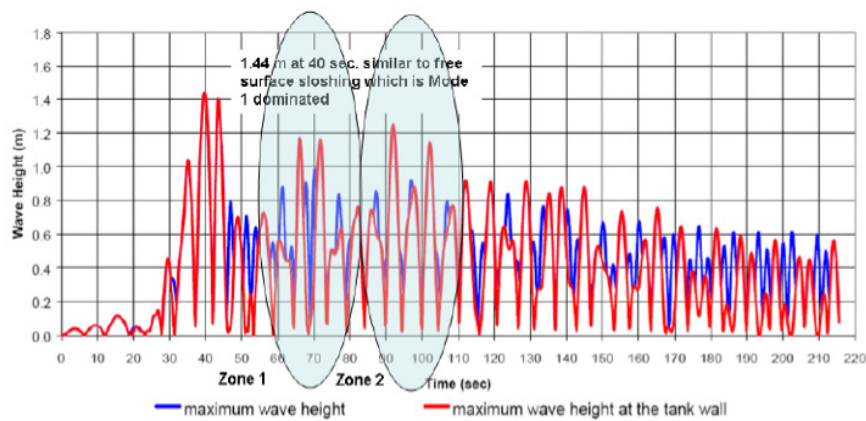
Figure 20 contains a plot of the maximum and minimum force per unit length in the exterior pontoon wall (outer rim) based on the results of the FSI solution. The lower right portion of the figure contains a comparison of the ExxonMobil CPA and the FSI solution at the time the maximum force occurs in the analysis (70.2 sec). The results agree very closely. Due to space restrictions, a limited amount of the results and comparisons are shown. Forces are used in the comparison due to their 1st order accuracy. In addition, the comparison of stresses based on two different Finite Element programs is difficult due to the differences in the element technology and mesh refinement.

Figure 21 contains a summary of the results obtained (in table) for the membrane forces and for the total thrust (circumferential) on the pontoon. The results from the two approaches are very close. There are significant differences in the complexity and sophistication of the two approaches. Some of the assumptions retained in the ExxonMobil CPA conform to principles and guidance in the Fire Service Law. The FSI analysis removes most of these assumptions.

The major differences in the two approaches are:

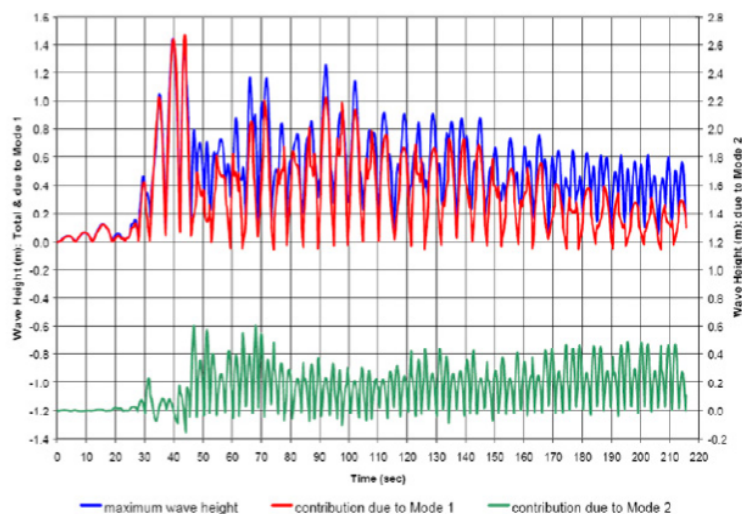
1. ExxonMobil CPA is designed to conform to the basic concepts embedded in the Japanese Fire Service Law. The goal of this work is to gain acceptance from the Japanese Fire Disaster Management Agency to use this approach in lieu of the simple spreadsheet solution. ExxonMobil determined that the spreadsheet approach was not consistently conservative for all tanks and overly conservative for many tanks. An over conservative calculation would result in costly and ineffective modifications to existing roofs.
2. The most significant difference between the ExxonMobil CPA and FSI is that the ExxonMobil CPA is based on the FSL Seismic Design Spectrum which limits the maximum velocity value to 210.8 cm/sec.
3. No modal phase relationships in the ExxonMobil CPA. FSI retains phase relationships.

Source: SIMULIA Community Conference, 2015



1. Zone 1 – Early Mode 1 response domination is first affected by growth in Mode 2 response contribution. Mode 1 asserts itself in Zone 2 until its contribution slowly dissipates. Time history analysis w/o the roof identical except zones shift forward slightly in time when roof present.
2. Zone 1 generates the most damage in pontoon as Mode 1 and Mode 2 compete for domination of the time history response.

Figure 18. FSI maximum wave height vs time.



1. Mode 2 contribution to maximum wave height reaches peak of 60 cm & a minimum of 28 cm over the time history.

Figure 19. Total maximum wave height - Mode 1 and Mode 2 contributions.

Source: SIMULIA Community Conference, 2015

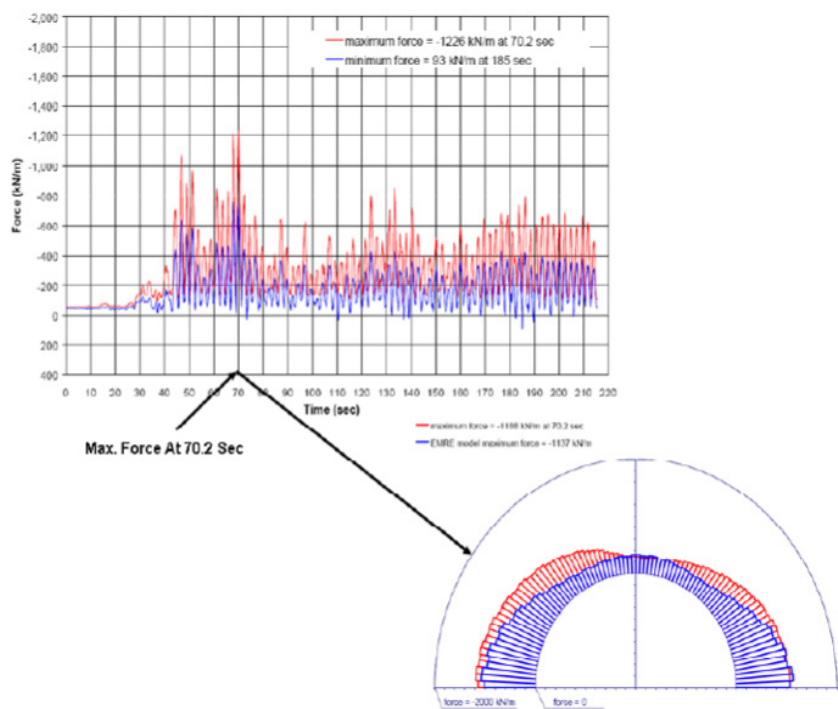


Figure 20. Compressive membrane force in exterior pontoon wall - FSI vs ExxonMobil CPA Method.

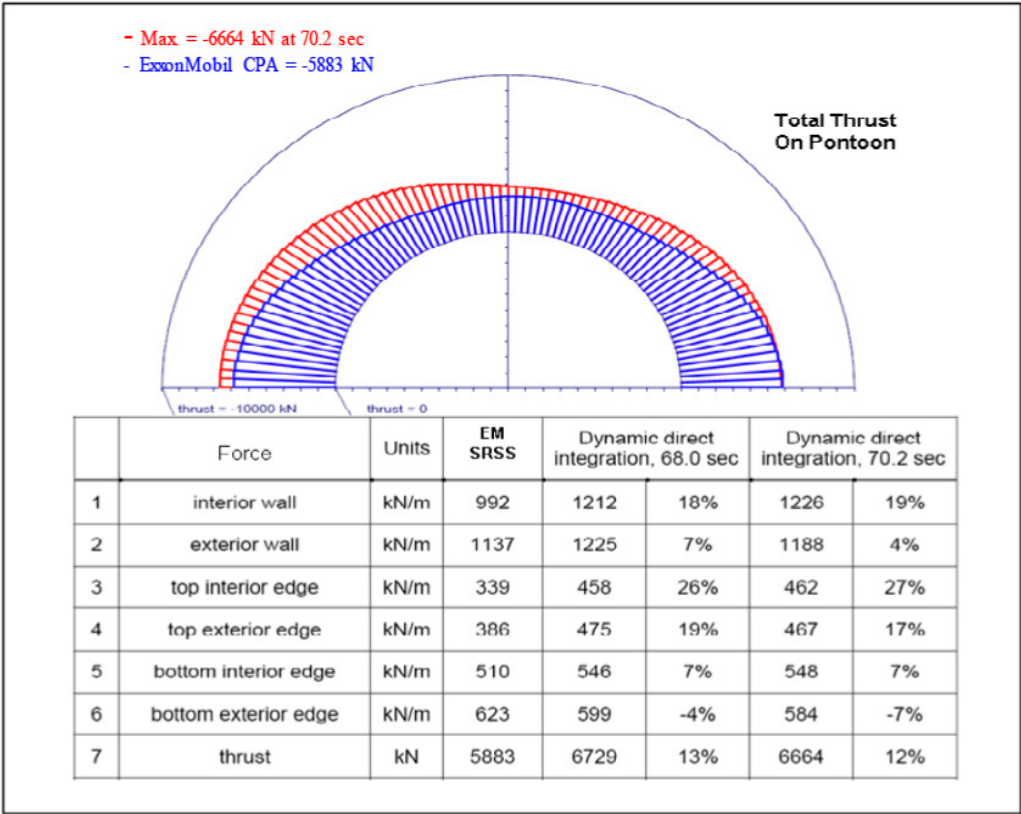


Figure 21. Sloshing force totals in pontoon components - ExxonMobil CPA vs time history analysis.

Source: SIMULIA Community Conference, 2015

For the example tank and floating roof, Mode 2 provides the dominant damage mechanism. The differences in the velocity spectrum values inherent in the two approaches can be approximately accounted for by scaling the ExxonMobil CPA results up to the FSI peak velocity spectrum value. Using a scaling factor of 1.14, the results in Figure 18 are adjusted and presented in Table 2.

Force	Units	EM SRSS	FSI At 68 Sec	FSI At 70.2 sec	Average FSI	Scaled EM	EM/FSI
Interior Wall	kN/m	992	1212	1226	1219	1117	92%
Exterior Wall	kN/m	1137	1225	1188	1207	1280	106%
Top Interior Edge	kN/m	339	458	462	460	382	83%
Top Exterior Edge	kN/m	386	475	467	471	435	92%
Bottom Interior Edge	kN/m	510	546	548	547	574	105%
Bottom Exterior Edge	kN/m	623	599	584	592	702	119%
Thrust	kN	5883	6729	6664	6697	6625	99%

Table 2. Scaled ExxonMobil CPA results to account for different velocity spectrum values.

CONCLUSIONS

1. ExxonMobil CPA represents a significant reduction in the required modeling and computational effort when compared to a dynamic direct integration fluid structure interaction analysis. It accomplishes this without a significant loss in accuracy.
2. ExxonMobil CPA provides adequate force and stress information to perform detailed evaluation of pontoon components and welds. This information enables structural modifications that improve structural integrity under seismic loading. This is particularly critical to evaluating the adequacy of welds.
3. ExxonMobil CPA uses a seismic design spectrum. FSI requires ground acceleration records and the results are only applicable to the corresponding earthquake. This limits the usefulness to post quake evaluation. CPA can be used for both design and analysis.
4. ExxonMobil CPA will enable the development of more comprehensive strain and buckling criteria for evaluating the structural integrity under seismic loading.
5. CPA is currently being extended to the issue of floatation stability due to the flooding of one or more pontoon compartments.

REFERENCES

1. Cacciatore, Philip J., "Seismic Analysis of Atmospheric Storage Tanks Using The ABAQUS Finite Element Program", ExxonMobil Report EE.256E.2004.
2. Cacciatore, Philip J., L. Magnus Gustafson, Yoichi Ishizaki, Koji Jingu, "Evaluation of the Seismic Response of Floating Roofs on Large Petrochemical Storage Tanks", ExxonMobil Report 2009 MEC 40, May 2009.
3. Guidelines for the Seismic Design of Oil and Gas Pipeline Systems, Committee on Gas and Liquid Fuel Lifelines of the ASCE Technical Council on Lifeline Earthquake Engineering, American Society of Civil Engineers, 1984.
4. Kikuchi, Tsutomu, and Masahiro Mura, "The Sloshing Simulation of Floating Roof Tank", PVP2005-71439, Proceedings of the 2005 ASME Pressure Vessels and Piping Division Conference, July, 2005
5. Kozak, Alexander and Ballard, Thomas A., "ExxonMobil Research and Engineering Company Seismic Analysis of a Liquid Storage Tank Floating Roof, SC Solutions Inc, Task Order No.: 620/4510270251, January, 2008
6. Report of Study to Develop Reasonable Modification Procedures to Satisfy Technical Standard Regarding Relatively Long Period Ground Motion for Facilities Handling Hazardous Materials, Japanese Government Fire Disaster and Management Agency, March 20006. (In Japanese).
7. Yamauchi, Yoshihiko, Kamei, Asamichi, Zama, and Yoshinori Uchida, "Seismic Design of Floating Roof of Oil Storage Tanks Under Liquid Sloshing", PVP2006-ICPVT11-93280, Proceedings of the 2006 ASME PVP Division Conference, July, 2006.

Source: SIMULIA Community Conference, 2015

ACKNOWLEDGEMENTS

As you might expect, a lot of ExxonMobil people made contributions to this work. In particular, the participation and support provided by the ExxonMobil Asia Area Engineering Office, including the efforts of Tatsuro Ohki-san, Yoichi Ishizaki-san and Koji Jingu-san have been greatly appreciated. The work done by Ishizaki-san in presenting the complex arguments of this work to the Japanese authorities provided immeasurable value to effort.

ExxonMobil would also like to acknowledge the contributions of Tsutomu Kikuchi, Senior Engineer at Idemitsu Kosan Co. Ltd. for his support and assistance in obtaining relevant tank data and damage photos, Thomas Ballard, formerly of SC Solutions Inc. and his work on the FSI solution, and Benjamin F. Hantz, formerly of ExxonMobil.

Source: SIMULIA Community Conference, 2015

Topology Optimization of a Flue Gas Ductwork with Significant Pressure-Loss Reduction

Background:

According to tightened emission regulations, energy suppliers are obliged to decrease steadily the NOX and COX emissions of their power plants. In order to meet NOX requirements with already exsistant power plants, a SCR (selective catalytic reduction) component may be added through a retrofit procedure. In that case, a low pressure drop in the SCR ductwork is required in order to guarantee a sufficient and efficient exhaust gas transport without an expensive replacement of the existing fan. E.ON Technologies GmbH, one of the major public utility companies in Europe, decided to check the potential of flow topology optimization via a service project, where the ductwork of a retrofitted SCR system was considered for optimization purposes. The objective of this project was to deliver two duct design proposals with low pressure loss and high flow uniformity.

Topology Optimization with SIMULIA Tosca Fluid

In a first step, E.ON Technologies GmbH provided constructional drawings of the SCR system, based on which a CAD-geometry was created. Afterwards, a CFD model was built to simulate the flow field inside of the existing flow ducts. The simulation results showed which sections of the duct system are suitable for a topology optimization – i.e. inflow duct and U-turn duct (see Fig.8).

In a next step, design space geometries were created for both sections. Those geometries represent the available design space E.ON Technologies GmbH could provide and are typical initial geometries for a topology optimization. The optimization was undertaken for the nominal mass flow rate, since the highest pressure losses were present for that (aerodynamic) load. After a runtime of 48 hours, the optimization reached a converged state. Based on the interpretation of the topology optimization results, a new CAD-design of the two design proposals was constructed (Fig.9). The displayed guide vanes were not part of the SIMULIA Tosca Fluid design proposal, but integrated manually during the reconstruction step.

The new designs revealed an impressive reduction in the total pressure loss for the design mass flow rate compared to the

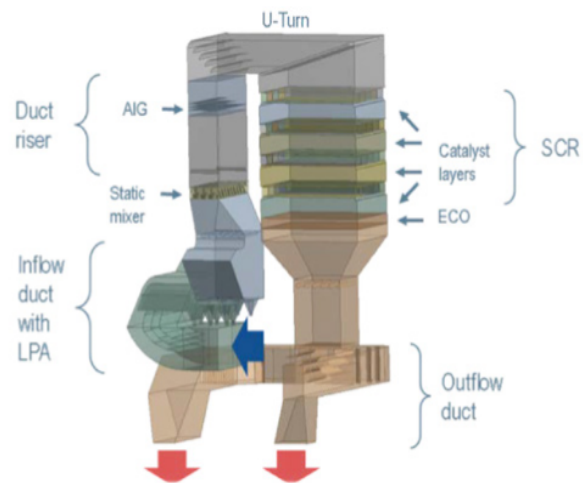


Figure 8 : CAD-model of existing duct design

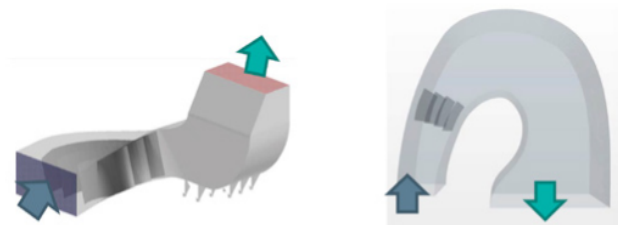


Figure 9 : Reconstructed designs for inflow duct (left) and U-turn duct (right)

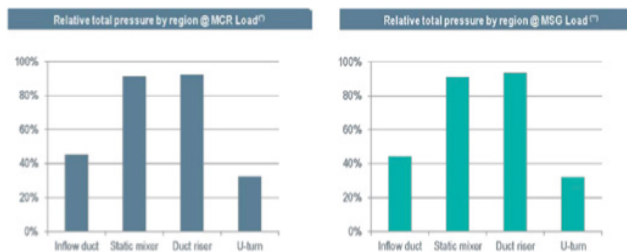


Figure 10 : Total pressure loss per section for design conditions (left) and reduced mass flow rate (right)

existing geometries labeled with 100% respectively (see Fig. 10). The validation run based on a reduced mass flow rate (called MSG load) showed that the optimized geometries have an improved loss behavior even for off-design conditions. Furthermore, by improving the loss behavior for inflow duct and U-Turn, even the flow losses within the non-modified sections (static mixer + U-turn) became smaller due to an overall homogenized flow field.

Excerpt from the whitepaper, "Simulation-based Fluid Optimization to Accelerate Development of Innovative Products"

Using Explicit Finite Element Analysis to Simulate Blast Loading on Hazardous Chemical Storage Tanks

Phillip E. Prueter (The Equity Engineering Group, Inc.)

Abstract: Accurately simulating the blast wave associated with an accidental explosion is extremely valuable in assessing the structural response and potential failure modes of critical process equipment such as storage tanks, piping, or pressure vessels in chemical and petrochemical facilities. Furthermore, assessing the potential damage from a blast wave can provide valuable information about protecting structures from external explosions and developing designs that improve blast damage mitigation. This paper discusses the underlying theory and examines the practical application of multiple finite element based explicit computational techniques for simulating the load acting on a structure due to a chemical explosion. Explicit three-dimensional blast analysis of single and double-walled storage tanks that carry an extremely high consequence of failure is performed. The structural response of the tanks due to postulated accidental explosions is investigated and likely failure modes are quantified and discussed.

The two explicit computational blast loading methods implemented in Abaqus/Explicit that are discussed and compared in this study are the incident wave loading model and CONWEP. The CONWEP model is appropriate for detonations of conventional explosions, and in this case automates many crucial features of the analysis, such as developing the overpressure time history, defining spatial decay, and accounting for reflection effects. Large vapor cloud explosions are known to behave differently than conventional explosives, and for these cases, the more manual incident wave loading approach permits the user to define their own time-varying overpressure amplitude. These advanced computational methods permit realistic evaluation of complex, three-dimensional process equipment subjected to accidental explosions.

Keywords: Explicit Finite Element Analysis, CONWEP, Blast Loading, Chemical Storage Tanks, Accidental Explosions, Blast Damage Mitigation, Abaqus/Explicit, Incident Wave Loading, Riks post-buckling analysis, TNT Equivalency

1. INTRODUCTION

A chemical explosion in air creates a blast wave as a result of the atmosphere surrounding the explosion source being pushed back. In general, the pressure of the compressed air at the blast wavefront decays as it moves away from the blast source. As discussed in many widely used books and publications (i.e. Smith and Hetherington, 1994, Kinney and Graham, 1985, Dusenberry, 2010, Bulson, 1997, ASCE, 1997, and DOD, 2008), a typical blast wave, as observed at a location removed from the center of a given chemical explosion, reaches a peak value of overpressure and begins to decay exponentially (as shown in Figure 1), eventually decreasing below atmospheric pressure. This peak overpressure decreases as the distance from the explosion source increases. The amount of time it takes the blast wavefront to reach a given point is known as the arrival time. The overpressure profile can be divided into the positive pressure portion (positive phase) and negative pressure portion (negative phase). Furthermore, the amount of time it takes the blast wave to decay below atmospheric pressure is known as the positive phase duration. The area under the curve during the positive phase duration is the positive blast wave impulse and is generally closely related to the damage capabilities of a given chemical explosion.

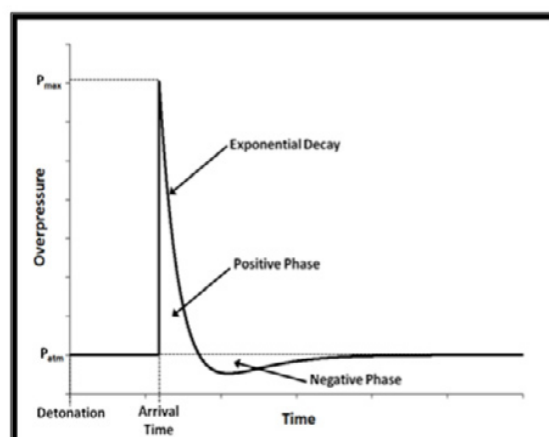


Figure 1. Typical overpressure amplitude for a chemical explosion (Simulia, 2010)

Source: SIMULIA Community Conference, 2014

Often times, the positive phase impulse is calculated and an equivalent triangular impulse with the same peak overpressure is used to approximate the realistic overpressure amplitude shown in Figure 1. In many cases, this approximation is appropriate because the positive phase portion of the incident pressure wave is typically the most damaging. Accurately simulating a realistic blast wave acting on a structure for a given explosive scenario is challenging because the peak overpressure, positive phase duration, amount of explosives, and distance from the explosion source all affect the overpressure amplitude. Furthermore, accounting for reflection effects on a loading surface adds further complication to the blast loading model; generally, structural response is fairly sensitive to capturing the correct incident wave reflection.

Two different Abaqus/Explicit finite element techniques used to simulate blast loading due to far-field, external chemical explosions in air are discussed herein; incident wave loading and the conventional weapons effects blast loading model, or CONWEP (Hyde, 1988). CONWEP is well suited for simulating the overpressure wave associated with the detonation of conventional explosives, while incident wave loading permits reasonable analysis of vapor cloud explosions. The underlying methodology of each approach is discussed in this paper as well as the advantages and applications of each. Furthermore, both computational methods are compared for the explicit dynamic simulations of realistic explosive overpressure waves (from a postulated concentrated explosion source and a vapor cloud) impacting in-service single and double-walled storage tanks that carry extremely high consequences of failure.

2. BLAST CURVE GENERATION

In order to develop the blast parameters required to generate realistic overpressure curves for conventional explosives, the equivalent TNT method is most commonly used. TNT equivalency of a given explosive is simply the ratio of the mass of TNT to the mass of explosive such that both yield equal pressure and impulse (Huntington-Thresher and Cullis, 2001). As discussed in (DOD, 2008), the ratio of energy output of a chemical explosive material relative to a given amount of TNT is calculated following

$$W_{effective} = \left[\frac{H_{exp}^d}{H_{TNT}^d} \right] W_{exp} \quad (1)$$

where $W_{effective}$ is the effective charge weight, W_{exp} is the weight of explosive in question, H_{exp}^d is the heat of detonation of a given explosive, and H_{TNT}^d is the heat of detonation of TNT. In order to compare shock wave parameters to available published data, a scaled distance from an explosion source to the loaded structure may be used. As discussed in (Baker, 1987), the most common form of blast scaling is the Hopkinson-Cranz or cube-root scaling method following

$$Z = \frac{R}{W^{(1/3)}} \quad (2)$$

where Z is the scaled distance, R is distance from the explosion source to the loading point, and W is the weight of TNT. This formulation implies that self-similar blast waves are produced at identical scaled distances when two different sized explosive charges (of similar geometry and chemical composition) are detonated in the same atmosphere. Using published empirical data (DOD, 2008), this scaled distance from the explosion source to a loading surface can be used to determine specific blast parameters. The CONWEP blast loading model discussed herein is based on a TNT explosion (assumed surface blast). Thus, when using CONWEP in Abaqus/Explicit, the mass of explosives in question is converted to an equivalent amount of TNT.

As outlined in (Tang and Baker, 1999), a major limitation associated with the equivalent TNT method is that the blast yield is a function of the amount of explosives and not the combustion mode; this makes this method less applicable when analyzing vapor cloud explosions. As discussed in numerous publications (Lees, 1996, Mercx et. al., 1998, Van den Bosch and Weterings, 1997, Woodward, 1998, and AIChE, 1994), vapor cloud explosions differ from conventional explosive detonations because flame accelerations, which can be exacerbated by confinement due to intervening structures, define the overpressure associated with a given explosion more so than simply the amount of fuel ignited. In order to quantify blast curves associated with vapor cloud explosions several widely accepted methods have been developed; TNO (Van den Bosch and Weterings, 1997) blast curves and Baker-Strehlow (Strehlow et. al., 1979) blast curves among others. In this study, triangular impulses of varying peak overpressures are used for simplicity when analyzing the structural response of a double-walled storage tank subjected to a postulated vapor cloud explosion (discussed below). This also permits evaluation of the structural response sensitivity to varying blast wave parameters, such as impulse duration.

Source: SIMULIA Community Conference, 2014

3. CONWEP BLAST LOADING AND INCIDENT WAVE LOADING

A significant challenge associated with computational blast modeling is appropriately accounting for reflection effects. The peak blast pressure that a structure is subjected to is greater than the peak incident overpressure and is referred to as the reflected pressure. This pressure is a function of the peak incident overpressure and the angle of incidence; that is, the angle between the loading direction and the surface normal of a given point on the loading surface. Generally, neglecting incident wave reflection entirely can yield non-conservative results, particularly for high peak overpressures, while assuming the maximum reflection effect at all points on a structure can potentially be overly conservative (especially for cylindrical geometries).

The CONWEP blast loading model is based on a collection of conventional weapons effects calculations from the equations and curves of TM 5-855-1 (Hyde, 1988) and is implemented into the Abaqus/Explicit through the CONWEP charge property (keyword *CONWEP CHARGE PROPERTY). The main advantage of using CONWEP over the incident wave loading approach is that realistic overpressure amplitudes (including both positive and negative phase) and other blast wave parameters are inherently calculated based on a user-defined amount of TNT at a given distance from the explosion source. Furthermore, the CONWEP blast loading model does not require modeling the fluid medium (air in this case) to account for reflection effects. The reflection formulation employed in the CONWEP charge property is depicted in Figure 2 below.

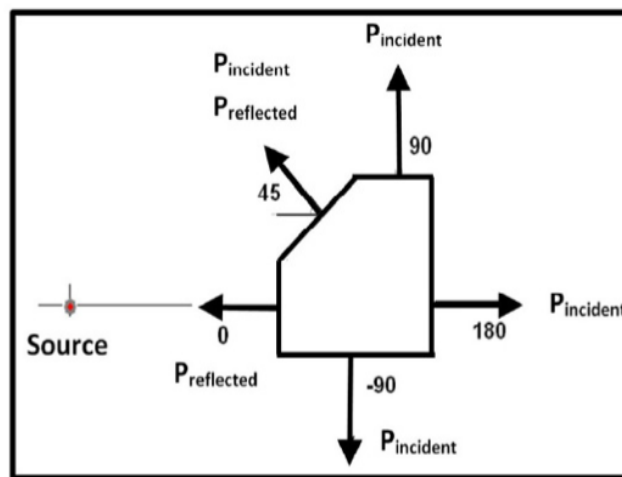


Figure 2. Blast wave reflection formulation built into the CONWEP charge property (Simulia, 2010)

In accordance with Figure 2 and the CONWEP formulation, the overpressure, as a function of angle of incidence and time ($P(t)$), observed on a loading surface is calculated as shown below:

$$P(t) = P_{incident}(t)[1 + \cos\theta - 2\cos^2\theta] + P_{reflected}(t)\cos^2\theta, \quad \text{for } \cos\theta \geq 0 \quad (3)$$

$$P(t) = P_{incident}(t), \quad \text{for } \cos\theta < 0$$

Incident wave loading is an explicit finite element technique in which an incident wave overpressure at a point can be defined as a function of space and time (Abaqus/Explicit keyword *INCIDENT WAVE INTERACTION). This overpressure amplitude is defined at a standoff point, located between the detonation point and the loaded structure. When simulating blast loading, realistic overpressure amplitudes need to be defined by the user. An incident wave can be defined as planar or spherical. A schematic showing the incident wave loading model in Abaqus/Explicit is given in Figure 3 below. For a more rigorous overview of the underlying theory behind both the CONWEP blast loading model and the spatial decay formulation employed in the incident wave loading model, refer to a recent computational study (Prueter et. al., 2012).

Source: SIMULIA Community Conference, 2014

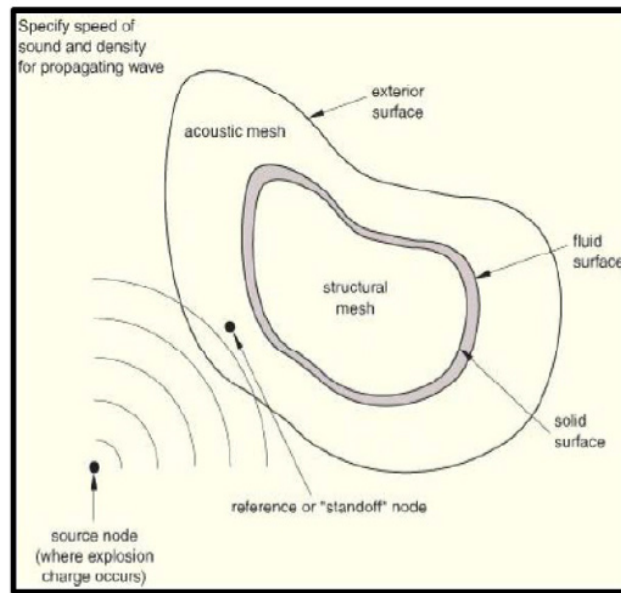


Figure 3. Incident wave loading model (Simulia, 2010)

In the incident wave loading model and as discussed in (Simulia, 2010), overpressure at a given point, x_j , can be defined as a function of space and time (t) following

$$P_I(x_j, t) = P_t(t)P_x(x_j), \quad P_x(x_j) = \left[\frac{R_o}{R_j} \right]^{\xi(R_j, R_o)} \quad (4)$$

where P_I is the incident pressure, P_t is pressure as a function of time, and P_x is pressure as a function of space. The pressure amplitude as a function of time, is defined by the user at a standoff point (keyword *AMPLITUDE), located between the detonation point and the loaded structure. An incident wave can be defined as planar or spherical. Spatial decay can be defined for spherical incident waves as shown above where R_o represents the distance between the source point and the standoff point and R_j represents the distance between the source point and some loaded point in space. The function $\xi(R_j, R_o)$ is used to define the spatial decay of a spherical incident wave.

4. EXPLICIT ANALYSIS OF A TANK SUBJECTED TO A VAPOR CLOUD EXPLOSION

As discussed in several case studies (Maremonti et. al., 1999 and Buncefield Investigation Board, 2008), the consequences of storage tank failure in chemical or petrochemical facilities can be catastrophic. To this end, it is estimated that 85 percent of all storage tank accidents are due to fire or explosion (Chang and Lin, 2006). Analyzing the structural response of storage tanks subjected to potential blast loading from conventional detonations or vapor cloud explosions is of great interest to engineers in the chemical and petrochemical industries. The explicit finite element analyses discussed in this paper represent examples of how computational blast simulation techniques can be applied to investigate the dynamic response of in-service process equipment.

The first computational blast analysis examined herein relates to a double-walled storage tank subjected to a postulated external vapor cloud explosion. A complete, elastic-plastic, three dimensional finite element model was constructed in Abaqus/CAE using linear shell elements (Abaqus S4 elements). Figure 3 shows the half-symmetry model with detailed geometric features.

Source: SIMULIA Community Conference, 2014

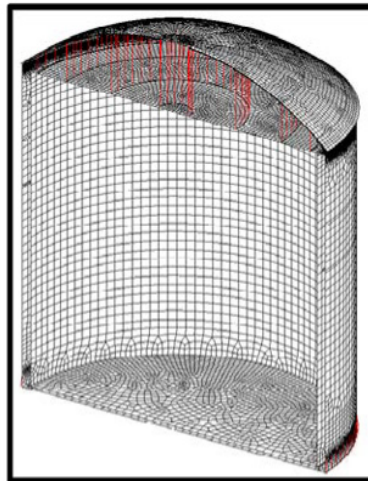


Figure 4. Vapor cloud explosion analysis 3D finite element model.

The baseline finite element model includes all major structural features, including the suspended deck, bracket supports, and the lower support straps. Quadratic beam elements are used to model the suspended deck hanger rods and the strap supports. Both the inner and outer tanks are modeled for completeness. Contact is defined between the suspended deck and the shell wall.

In this case, because of the range of potential vapor cloud explosive scenarios, iterations are performed using incident wave loading to determine the maximum permissible overpressures and loading durations that predict containment and safe operability. The focus is on establishing a blast rating for the structure, independent of a specific explosion scenario. This provides an essential characteristic of the structure that can be referred to for any future blast loading scenarios. Peak reflected overpressures of 5, 8, and 11 psi are applied for impulse duration of 50 milliseconds to the model. Figure 5 compares maximum deflections and stresses in the domed roof (opposite impact side) for these varying peak overpressures. This figure demonstrates the relatively constant nature of the largest displacements. In the 5 psi case, maximum deflections are generally less than 12 inches with relatively little accumulation in plastic strain.

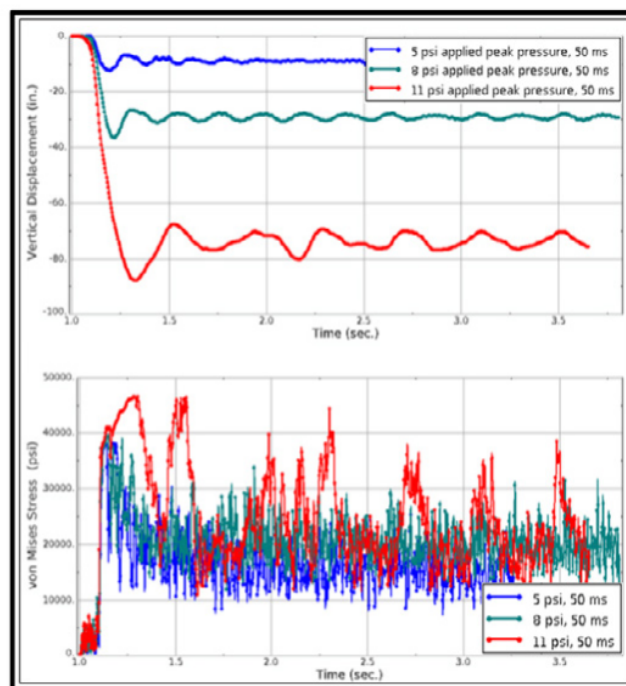


Figure 5. Deflection/stresses in domed roof opposite of impact side for varying peak overpressures

Source: SIMULIA Community Conference, 2014

These results indicate that a peak reflected overpressure of 5 psi is near the limit to maintain tank containment following blast loading. Furthermore, for the 8 and 11 psi overpressure cases, results indicate that plastic collapse of the domed roof and tank shell would be likely, and containment of the tank contents would likely be jeopardized. This analysis also indicates that the suspended deck is a critical feature of the tank, and is also an indicator of safe operability. The suspended deck is one of the weakest points of the tank internals; results from the series of analyses show that a maximum peak reflected pressure of 2 psi is advisable to ensure safe tank operability (again, impulse duration assumed to be 50 milliseconds). This corresponds to a maximum side-on overpressure of 1 psi with a uniform/constant reflection coefficient of 2.

Additionally, in order to compare the effect of varying blast wave impulse durations, additional iterations are performed where overpressure is held constant and impulse duration is changed. As expected, the duration of the impulse has a significant effect on the overall structural damage. Lower peak overpressures acting at longer durations could also be damaging. Because of the variability in overpressure profiles from vapor cloud explosions, incident wave loading is employed, and offers the flexibility to perform parametric studies for different blast loads.

5. EXPLICIT ANALYSIS OF A DOUBLED-WALLED STORAGE TANK SUBJECTED TO A CONVENTIONAL EXPLOSION

A second double-walled storage tank that carries a high consequence of failure is analyzed; this time the postulated blast loading is due to the accidental detonation of a known amount of concentrated, combustible fuel. Again a complete, elastic-plastic, three dimensional finite element model is constructed in Abaqus/CAE using linear shell elements (Abaqus S4 elements). Figure 6 shows a cutaway view of the model with contours of shell element thickness, and shows the detail of the outer tank roof support structure (also modeled using linear shell elements).

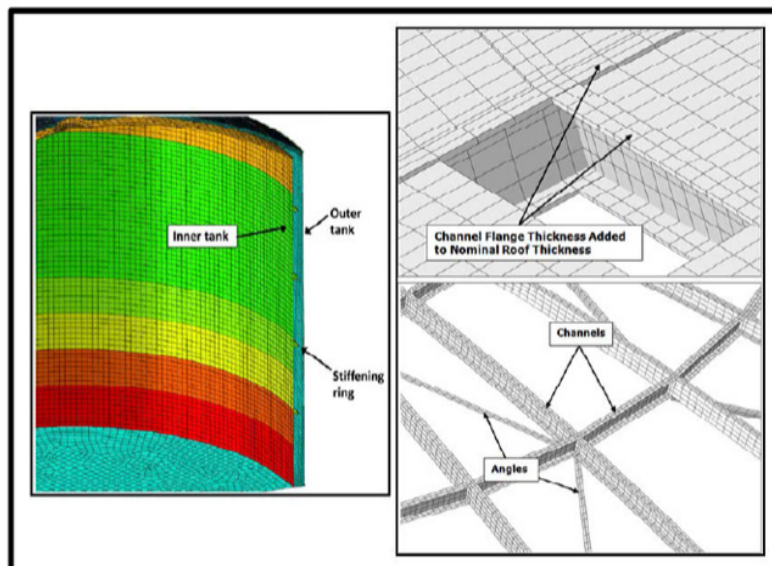


Figure 6. Cutaway view of the complete finite element model with shell thickness contours shown (left) and outer tank roof support structure details (right)

The inner tank has four external stiffening rings that extend into the annular space between the inner and outer tank. The outer tank roof support structure consisting of interconnected channels and angles is also modeled in detail (see Figure 6), and is included in the finite element model. Contact is defined between the tank structures and the effects of self-weight are included.

An initial analysis is performed on just the outer tank (unstiffened shell) to investigate the plastic strains and wall deflections due to the applied blast wave. Both the CONWEP (assumed surface blast) and incident wave loading models are compared. Due to the size of the finite element model and the significant computational requirements, the fluid medium is not explicitly modeled for the incident wave simulation. An overpressure amplitude that accounts for reflection effects is used; that is, the peak reflected overpressure is applied to the entire incident wave model. Furthermore, in order to mimic the behavior of the CONWEP simulation

Source: SIMULIA Community Conference, 2014

as much as possible, the overpressure profile defined for the incident wave simulation as a function of time does, in fact, consist of positive and negative phases and decays exponentially (obtained from a CONWEP simulation on a single, fixed element). The side-on reflection coefficient for this overpressure magnitude is 2.

Even for the relatively small peak reflected overpressure of roughly 1.9 psi, the outer tank wall deflections prove to be significant, to the point of local collapse. An Abaqus/Standard static post-buckling analysis (using the Riks solver), with surface pressure applied using an analytical field definition to match surface pressure in the dynamic simulations confirms that the outer tank wall buckles before the peak reflected pressure is reached. Riks post-buckling analyses are discussed in more detail below.

For comparison, Figure 7 shows contours of outer tank plastic deformations (identical scale) due to the overpressure wave for blast loading from the CONWEP charge property and incident wave loading.

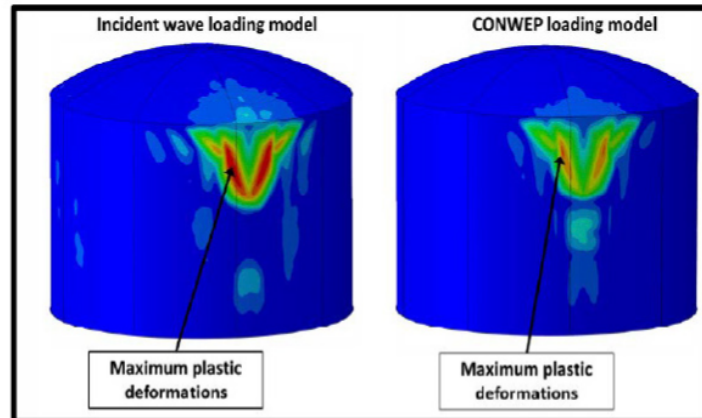


Figure 7. Outer tank plastic deformations due to blast loading using the CONWEP and incident wave loading models (contour scales identical)

As expected, the incident wave approach produces higher deflections than CONWEP because the applied peak reflected overpressure does not decrease as angle of incidence increases (moving around the circumference of the tank). It is noted that the general agreement between methods is generally quite good. This suggests that the simplification of a single maximum reflection coefficient in this case for the incident wave approach is not excessively conservative, and that reasonable results can be obtained without modeling the entire acoustic domain, in which case reflection effects would be more rigorously accounted for (may not hold true for higher pressures).

For the blast analysis of the complete finite element model that includes both the inner and outer tanks with contact defined between shells, the outer wall deflections are much less significant because the outer tank impacts the inner tank stiffeners. Essentially, the inner tank stiffening rings absorb the impact from the outer tank wall and plastically deform. The resulting deflections in the inner tank shell are fairly minimal. This is demonstrated in Figure 8, which shows the maximum deflection during blast loading from the CONWEP simulation of the inner tank and the outer tank. The contour scale for the inner tank corresponds to 10 percent of the outer tank contour scale, and the maximum inner tank wall deflection is less than 10 percent of the maximum outer tank wall deflection. The maximum outer tank deflections occur in the regions in between the inner tank stiffening rings, and the outer tank shell wall never directly impacts the inner tank shell wall. These relatively minimal inner tank deflections suggest that the sloshing effects of the liquid contained by the inner tank would likely be minimal.

Source: SIMULIA Community Conference, 2014

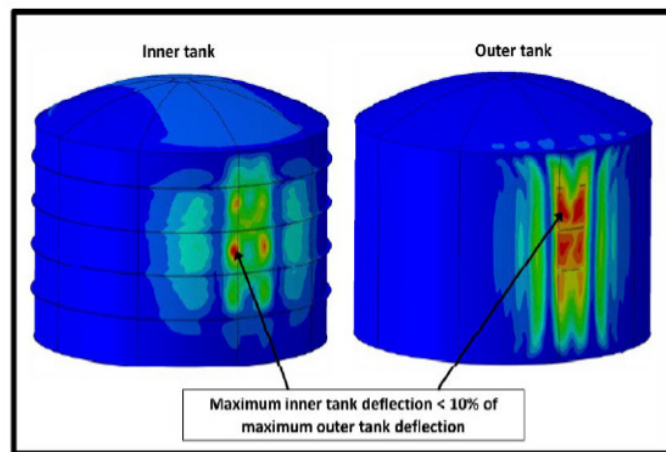


Figure 8. Maximum deflections of the inner and outer tanks due to CONWEP blast loading (inner tank contour scale corresponds to 10 percent of outer tank scale)

These results indicate that from a blast damage mitigation perspective, the four inner tank stiffening rings prove to be very beneficial in reducing the severity of the impact between the inner and outer tanks. While both explicit computational techniques used to analyze this double-walled storage tank give reasonably similar results, from a convenience standpoint, CONWEP proves to be advantageous for conventional detonations because the appropriate overpressure and spatial decay are automatically calculated for the user-defined equivalent amount of TNT. Additionally, due to the built-in formulations that account for overpressure reflection effects, some conservatism is removed in the CONWEP case compared to incident wave loading, making it the natural choice for well-defined, conventional explosive detonation scenarios.

6. EXPLICIT ANALYSIS OF SINGLE-WALLED STORAGE TANKS

The postulated blast source in this case is the same source as discussed in the previous example (conventional detonation). However, this cluster of single-walled tanks is located much closer to the blast source, meaning peak overpressures are much higher than the above example. These tanks have self-supporting roofs and are anchored to their respective foundations via hold-down clips and hold-down bolts. As is the case for the above mentioned storage tanks, these tanks carry an extremely high consequence of failure due to their process contents.

Three dimensional, elastic-plastic finite element models of each tank are constructed in Abaqus/CAE using linear shell elements (Abaqus S4 elements) and explicitly analyzed using CONWEP. A node at the center of each hold-down clip is fixed in all degrees of freedom and then coupled to the surrounding bolt hole (keywords *KINEMATIC and *COUPLING) to simulate the restraint of the bolts as shown in Figure 9. A rigid surface representing the tank foundation is constructed and included in the model, and contact between this rigid surface and the tank bottom is defined. This ensures that the tank will not unrealistically sag due to gravity alone and penetrate the foundation due to self-weight (inducing non-realistic stresses in the tank floor and shell). This modeling technique also permits the tank bottom to physically lift off its foundation when the blast load is applied. In order to account for the weight of the process fluid, an equivalent pressure is applied to the tank bottom that corresponds to the weight of fluid. This pressure is applied throughout the simulation, implying that sloshing effects are neglected.

Source: SIMULIA Community Conference, 2014

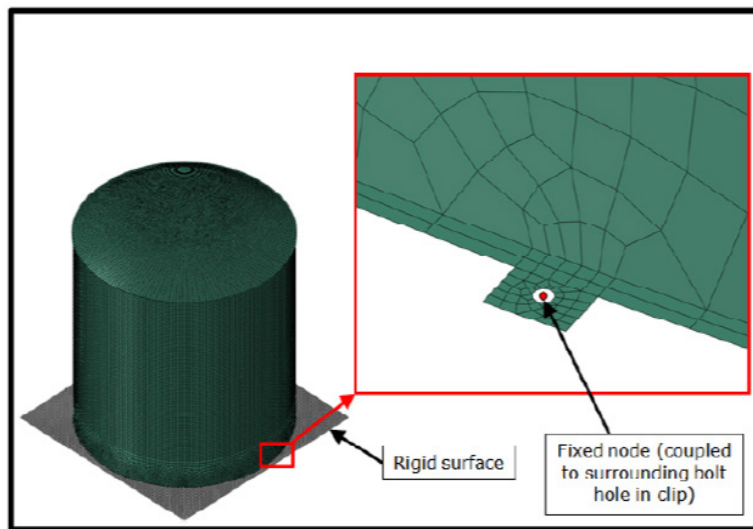


Figure 9. Finite element model of a single-walled storage tank

In this case, after the onset of the blast wave, the wall and the roof of each tank significantly deform. In order to better visualize the tank behavior following the onset of the blast wave impact, Figure 10 shows contours of deflection as time elapses for the tank closest to the blast source (time scale given is seconds after detonation). These large deflections predicted in the shell and roof of the tank are more indicative of plastic collapse-like behavior than a dynamic or vibratory failure. Furthermore, the stresses observed in the tank shortly after blast wave impact are significant and support the prediction of a plastic collapse failure mode.

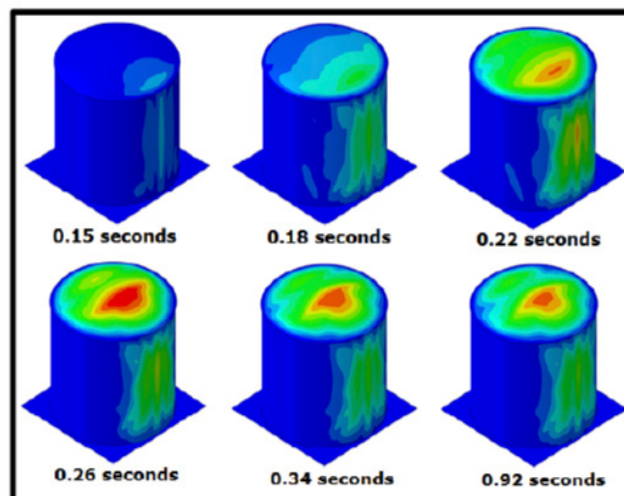


Figure 10. Progression of deformation due to blast loading in Abaqus/Explicit

Additionally, the stresses in the tank at the hold-down clips and reaction forces at the assumed bolt locations predict likely failure of either the bolts themselves or the hold-down clips. Figure 11 shows equivalent plastic strains in the tank closest to the detonation source after the impact of the blast wave (contour scale capped at 5%). The hold-down clips and other portions of the tank wall near the connection with the hold-down clips exceed 100% equivalent plastic strain. Results for the other tanks in the cluster yield similar stresses and plastic strains. In order to further investigate the possibility of plastic collapse or buckling, static Riks post-buckling analyses are performed on each tank using Abaqus/Standard. A static pressure distribution is applied to mimic peak

Source: SIMULIA Community Conference, 2014

blast loading pressures. This is achieved by defining an analytical pressure distribution in Abaqus/CAE as a function of the global coordinates (higher order polynomial functions are defined in multiple directions).

A Riks analysis is useful for simulating non-linear instability problems, and includes the effects of large deformations and elastic-plastic material behavior. It can also be used to look at post-buckling or collapse shapes. The applied load in a Riks analysis is actually a variable and part of the solution for each increment, and is always relative to the current deformed configuration of the structure. If no converged solution is possible for an incremental increase in load, then the equilibrium solutions can be calculated with a decreased applied load. This is the mechanism that allows post-buckled shapes to be predicted and visualized. In extreme cases, no equilibrium solution can be reached, even with decreased load, and only load reversal is predicted to lead to a stable solution. This is the case for the single-walled tanks discussed herein, where once collapse has initiated, displacements are predicted to be essentially unbounded for any amount of positive load and thus, surface pressure becomes negative before peak overpressure values are achieved (in this case, load reversal occurs at a fraction of peak reflected overpressure).

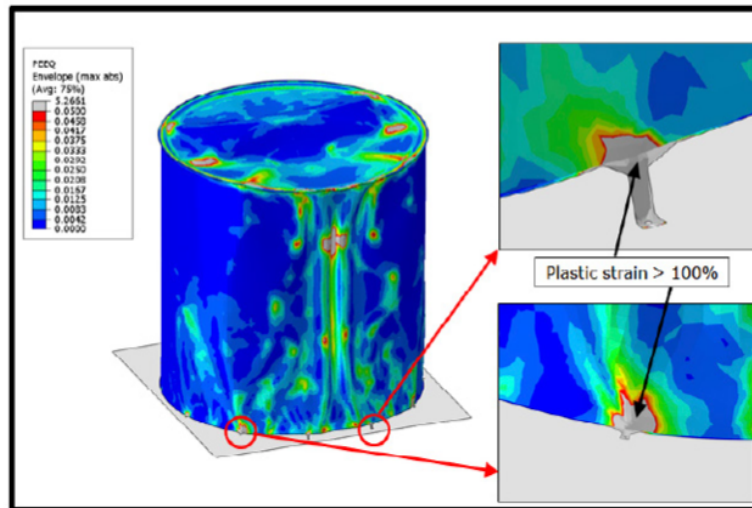


Figure 11. Equivalent plastic strain due to blast loading (contour scale capped at 5%)

Based on the computational results discussed above, the three storage tanks subjected to the assumed postulated blast load are likely to fail via several possible failure modes. The first likely failure mode is plastic collapse of the tank walls and/or roofs. All three tanks see large deflections, and Riks post-buckling analyses confirm that the magnitude of deflections seen in the tank walls would likely result in unstable collapse behavior before the peak blast overpressure is achieved. A second likely failure mode is extreme damage in the tanks at the hold-down bolts and/or regions of the tanks that are connected to the bolts. In all three tanks, equivalent plastic strains exceed 100% in these regions.

Failure of multiple hold-down bolts or hold-down clips could cause gross failure of tank anchorage, and as a result, the tank could slide off its foundation or potentially overturn. Furthermore, sloshing effects of the fluid inside the tanks are neglected in the simulations discussed herein, but would possibly increase the likelihood of failure and possibly contribute to tank collapse or overturning. Essentially, the close proximity of all three tanks to the postulated blast source and corresponding blast wave overpressures indicate that failure would be probable if the given detonation event were to actually occur. Design changes to the tanks to supplement blast damage mitigation would likely have to be drastic in order to achieve a significant reduction in the overall probability of failure.

The analyses discussed herein employ explicit FEA formulations, which in general, are well suited for analyzing highly transient, non-linear behavior and permit evaluation of models with highly discontinuous events with large deformations. Explicit FEA simulations are computationally expensive due to the small time increments generally needed to achieve convergence. It is noted that in order to achieve practical simulation times, all of the explicit analyses discussed herein are conducted across multiple CPUs (often 16 or more) simultaneously. Utilizing parallel computing permits explicit evaluation of complex, three dimensional structures, such as the storage tanks discussed in this paper. Further details of the explicit FEA performed on the hazardous chemical storage tanks discussed herein are available in a recent publication (Prueter and Dewees, 2013).

Source: SIMULIA Community Conference, 2014

7. CONCLUSIONS

This study compares two different Abaqus/Explicit computational methods for modeling blast loading on structures; incident wave loading (Abaqus/Explicit keyword *INCIDENT WAVE INTERACTION) and the CONWEP charge property (Abaqus/Explicit keyword *CONWEP CHARGE PROPERTY). The conventional weapons effects blast loading model (CONWEP) is well suited for conventional detonations and uses the equivalent TNT method. It also inherently calculates the appropriate blast wave parameters for a given detonation event. Additionally, CONWEP achieves realistic overpressure amplitudes, spatial decay, and reflection effects without having to model the fluid medium, as is the case for incident wave loading. Incident wave loading provides the user with the flexibility to define time-varying overpressure amplitudes. This is useful when performing iterative or sensitivity studies to evaluate blast loading parameters or when attempting to characterize the blast wave associated with a vapor cloud explosion that does not behave like a conventional detonation. The development and implementation of the CONWEP and incident wave blast loading models into Abaqus/Explicit makes accurately analyzing the response of complex, 3D structures subjected to blast loading feasible.

Additionally, explicit, three dimensional finite element analyses of both single and double-walled storage tanks that carry an extremely high consequence of failure are discussed herein. Incident wave loading is used iteratively to determine the maximum permissible overpressure for a double-walled storage tank subject to a postulated vapor cloud explosion. The CONWEP blast loading model is used to analyze single and double-walled storage tanks subjected to a postulated chemical explosion (conventional detonation of concentrated chemical explosives). Gaining a better understanding of the possible failure modes due to blast loading of critical process equipment in the petrochemical and related industries is very beneficial to engineers working in these fields. The explicit computational methods built into Abaqus/Explicit, such as the ones examined herein, can provide valuable insight into the blast resistance of existing structures and can guide design decisions regarding how to mitigate potential blast damage on fixed process equipment.

8. REFERENCES

1. AIChE, "Guidelines for Evaluating the Characteristics of Vapor Cloud Explosions, Flash Fires, and BLEVEs," Center for Chemical Process Safety of the American Institute of Chemical Engineers., New York, 1994.
2. ASCE, "Design of Blast-Resistant Buildings in Petrochemical Facilities, 1st ed," Task Committee on Blast-Resistant Design, American Society of Civil Engineers., Reston, Virginia, 1997.
3. Baker, W., "Design Considerations for Toxic Chemical and Explosives Facilities – Chapter 1, Blast Pressure Effects: An Overview," American Chemical Society, 1987.
4. Bulson, P., "Explosive Loading of Engineering Structures, 1st ed," Taylor and Francis Group, London, 1997.
5. Buncefield Investigation Board, "The Buncefield Incident 11 December 2005: The Final Report of the Major Incident Investigation Board," Volume 1, 2008.
6. Chang, J., and Lin, C., "A Study of Storage Tank Accidents," Journal of Loss Prevention in the Process Industries, 19(1) 51-59, 2006.
7. DOD, "Unified Facilities Criteria (UFC) 3-340-02: Structures to Resist the Effects of Accidental Explosions," Department of Defense, United States of America, 2008
8. DSS Simulia, Abaqus 6.10-1, "Analysis Users' Manual," 2010.
9. Dusenberry, D., "Handbook for Blast-Resistant Design of Buildings, 1st ed," John Wiley and Sons, Inc., Hoboken, New Jersey, 2010.
10. Huntington-Thresher, W., and Cullis, I., "TNT Blast Scaling for Small Charges," 19th International Symposium of Ballistics, 2001.
11. Hyde, D., "Users' Guide for Microcomputer Programs CONWEP and FUNPRO – Applications of TM 5-885-1," U.S. Army Engineer Waterways Experimental Station, Vicksburg, VA, 1988.
12. Kinney, G., and Graham, K., "Explosive Shocks in Air, 2nd ed," Springer-Verlag, Berlin, 1985.
13. Lees, F.P., "Loss Prevention in the Process Industries," Butterworth-Heinemann, Oxford, 1996.
14. Maremonti, M., Russo, G., Salzano, E., and Tufano, V., "Post-Accident Analysis of Vapor Cloud Explosions in Fuel Storage Areas," Trans IChemE, 77(B), 1999.
15. Mercx, W.P.M., van den Berg, A.C. and van Leeuwen, D., TNO report PML 1998-C53, "Application of correlations to quantify the source strength of vapor cloud explosions in realistic situations, Final report for the project: GAMES," Netherlands Organization for Applied Scientific Research (TNO), 1998.
16. Prueter, P.E. and Dewees, D.J., "Using Explicit Finite Element Analysis to Simulate the Effects of External Chemical Explosions on

Source: SIMULIA Community Conference, 2014

- Single and Double-Walled Storage Tanks," In Proceedings of the AIChE Spring Meeting and 9th Global Congress on Process Safety, San Antonio, Texas, 2013.
17. Prueter, P.E., Dewees, D.J., and Brown, R.G. "Examining Computational Techniques to Simulate the Effects of External Chemical Explosions Using Explicit Finite Element Analysis," In ASME Pressure Vessels and Piping Conference. PVP2012-78566, 2012.
 18. Smith, P., and Hetherington, J., "Blast and Ballistic Loading of Structures, 1st ed," Butterworth-Heinemann, Oxford, 1994.
 19. Strehlow, R.A., Luckritz, R.T., Adamczyk, A.A., and Shimpi, S.A. "The Blast Wave Generated by Spherical Flames," Combustion and Flame, Vol. 35, 297-310, 1979.
 20. Tang, M.J., and Baker, Q.A., "A New Set of Blast Curves from Vapor Cloud Explosion," Process Safety Progress. Vol. 18, No. 3, 1999.
 21. Van den Bosch, C.J.H. and R.A.P.M. Weterings, eds., "Methods for the calculation of physical effects: due to releases of hazardous materials (liquids and gases) ("TNO Yellow Book"), CPR 14E, Parts 1 and 2, 3rd Edition," Director-General for Social Affairs and Employment, Committee for the Prevention of Disasters. The Hague, 1997.
 22. Woodward, J. L., "Estimating the Flammable Mass of a Vapor Cloud," Center for Chemical Process Safety of the American Institute of Chemical Engineers, New York, 1998.

Source: SIMULIA Community Conference, 2014

Air Blast Loaded Safety Device Dynamical Response Analysis

Tropkin S.N., Tlyasheva R.R., Bayazitov M.I., Rafikova Z.R., Kuzeev I.R. (Ufa State Petroleum Technological University)

Abstract: Equipment of many oil and gas processing plants in Russian Federation is considerably worn-out. This cause the decrease of reliability and durability of equipment, and to rise of accident rate. An air explosion is the one of the most dangerous cases for plants in oil and gas industry, usually caused by uncontrolled emission and inflammation of oil products. Air explosion can lead to significant danger for life and health of plant staff, so it needs necessity of safety device usage. New type of safety device designed. Numerical simulation is necessary to analyze design parameters and performance of safety device, subjected to air blast loading. Coupled fluid-structure interaction analysis performed to determine strength of protective devise and its performance. Coupled Euler-Lagrange method allowable in Abaqus by SIMULIA selected as the most appropriate analysis tool to study blast wave interaction with safety device. Absorption factors of blast wave evaluated for safety devise. This factors allow to assess efficiency of safety devise, and its main structural component – dampener. Usage of CEL allowed to fast and accurate model the dampener behavior, and to develop the parametric model to determine safety devise sizes.

Keywords: deflagration, air blast wave, finite element method, internal energy, deformation, safety devise, Coupled Euler-Lagrange method

1. INTRODUCTION

Equipment of many chemical, petrochemical, oil and gas processing plants in Russian Federation is considerably worn-out. Besides, equipment exploitation regimes very often disrupted by hold-up in work materials delivery, low technological and manufacturing discipline etc. This factors cause decrease of reliability and durability of equipment, and rise of accident rate. In current economic situations, companies forced to use equipment until it partial of full outage.

Any accident on chemical, petrochemical, or oil & gas processing plant can cause a harm of personnel involved in production, harm the nearby plants personnel and civilians. This feature distinguish chemical and oil & gas plants from other industries plants, which usually do not cause risk to surrounding objects in case of extreme situation. Thereby, due to high potential danger of petrochemical and oil & gas plants, the problem of safeguarding for such objects is actual.

An air explosion is the one of the most dangerous cases for plants in oil and gas industry, usually caused by uncontrolled emission and inflammation of oil products. Very often constructions used in oil plants cannot sustain such disaster, because they designed without consider of air explosion loading or became depreciated. Air explosion can lead to significant danger for life and health of plant staff, so it needs necessity of safety device usage.

Usually that type of construction is operating station, which controls most of processes at oil plant, and where most of plant staff are present. Reconstruction of such a stations required great amount of time, and could bring financial losses due to the plant idle. In such case, most reasonably is using protective devices instead of reconstruction. Air explosion danger for such buildings as operation stations, may be significantly reduced by corresponding safety device.

2. SAFETY DEVICE FEATURES

Team of authors developed new type of protective device, which are able significantly reduce shock wave front pressure excess and thus reduce blast loading for operational station [1].

Safety device include special dampers, mounted on frame made from I-beams. Devise designed to disseminate the blast wave energy by partial reflection and scatter in the extinguishers.

Damper designed as welded box with specially oriented steel plates inside, which shape is comb-like channel inside the box. This comb-like channel cause the blast wave reflection between plates and decrease the overpressure. Damper also can decrease wave energy by its dissipation to elastic-plastic deformation due to plate collapse. This two energy dissipation mechanism allow to gradually decreasing wave overpressure. Safety devise sketch is shown on Figure 1.

Source: SIMULIA Community Conference, 2014

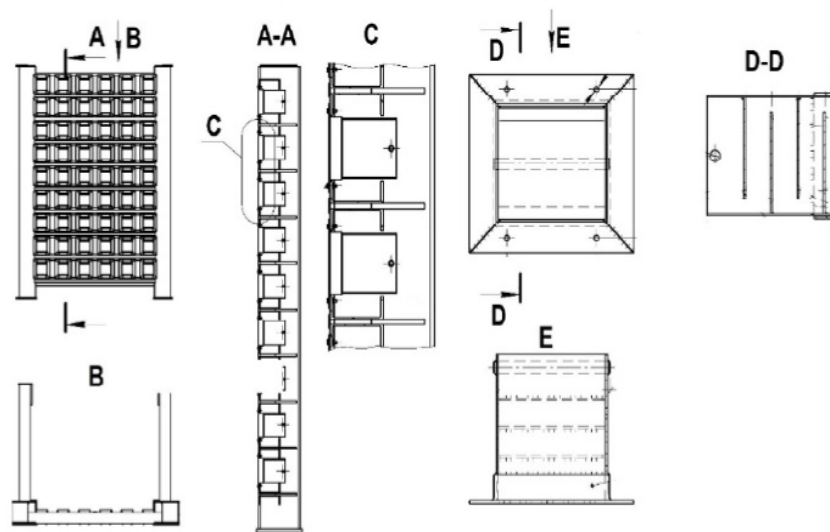


Figure 1. Safety device and damper sketch.

3. RESEARCH METHOD

To analyze safety device behavior under blast load we must solve coupled flow dynamic and dynamic stress problems. Analysis of such problem with analytical methods is complicated due to necessity of solution of large amount of coupled nonlinear problems, and an empirical method is complicated due to cost of experiments involving blast waves.

To find such coupled solution, numerical simulation model need to be made. Many commercial solvers can solve coupled solution of fluid structure interaction. In this study, we have used the CEL technology in Abaqus/Explicit.

One of most convenient techniques to analyze fluid-structure interaction problems is to use Coupled Euler Lagrange analysis method, which implemented in Abaqus/Explicit. This method allows simultaneously model coupled dynamic flow and stressing problem in single solution space.

It is necessary to determine efficiency of protective device due to blast waves loading with different parameters. In addition, it is necessary to determine deformation or collapse behavior of damper – its deformation behavior directly affects the efficiency of shock wave energy reduction.

Simulation for such analysis made with Abaqus/Explicit solver, and divided to the following steps:

1. Flow simulation of blast wave propagation through the media.
2. Dynamic stress simulation of the damper.
3. Coupled fluid-structure interaction simulation of blast wave propagation through the deformable extinguisher.

Step 1 and 2 necessary to verify fluid and dynamic stress simulation. Coupled model completely assembled after such verification. Combination for Coupled Euler-Lagrange analysis shown on Figure 2.

Source: SIMULIA Community Conference, 2014

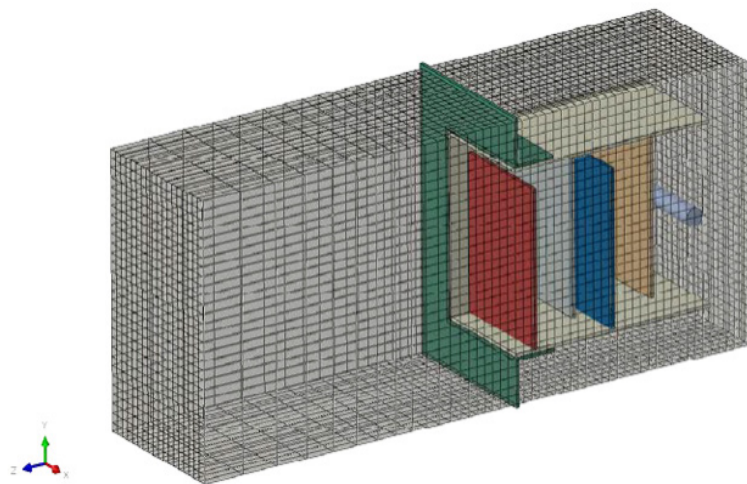


Figure 2. Coupled Euler-Lagrange simulation model.

General contact technology used to transfer pressures and mesh motions between flow and dynamic strength analysis. This technology allow to automatically tracking interaction surfaces in simulation media.

Is result of simulation achieved the pressure distribution in gas media and plastic deformations in extinguisher.

4. RESEARCH RESULTS

For the analysis of safety device and its extinguishers blast wave parameters defined under the regulation methodic [2] and [3]. Risk class 3 objects containing 5000 kg of gasoline selected for analysis. Calculated deflagration wave parameters shown on Figure 3.

Figure 3. Blast wave parameters

	R	E (J)	$\Delta P+$	$\Delta P-$	T+ (s)	T- (s)
5000 kg	50	2,2E+11	198 KPa	30 Kpa	0,0616	0,204

Where R – distance from explosion center, E – effective explosion energy, $\Delta P+$ - positive wave peak overpressure, $\Delta P-$ – negative wave pressure, T+ - positive wave duration, T- negative wave duration.

Pressure and velocity distribution in gas media shown on Figure 4 and 5. Pressure distribution diagram for different points in extinguisher shown on Figure 6.

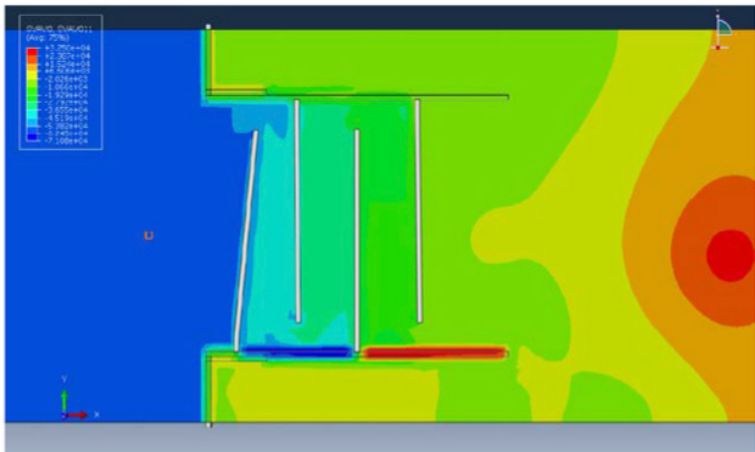


Figure 4. Pressure distribution over deformed damper.

Source: SIMULIA Community Conference, 2014

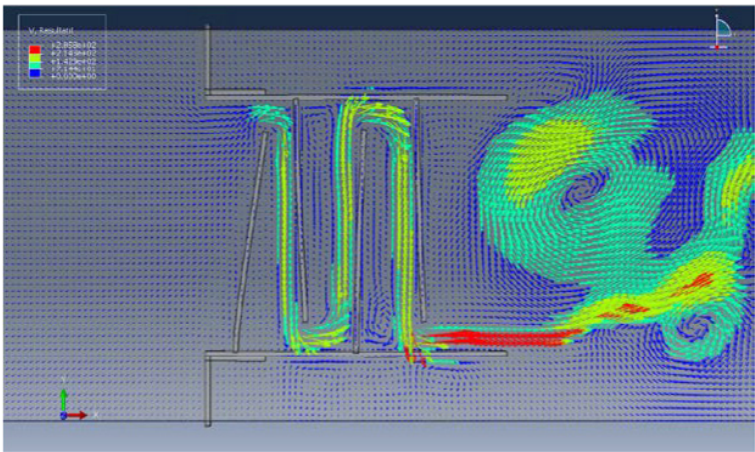


Figure 5. Flow velocity distribution over deformed damper.

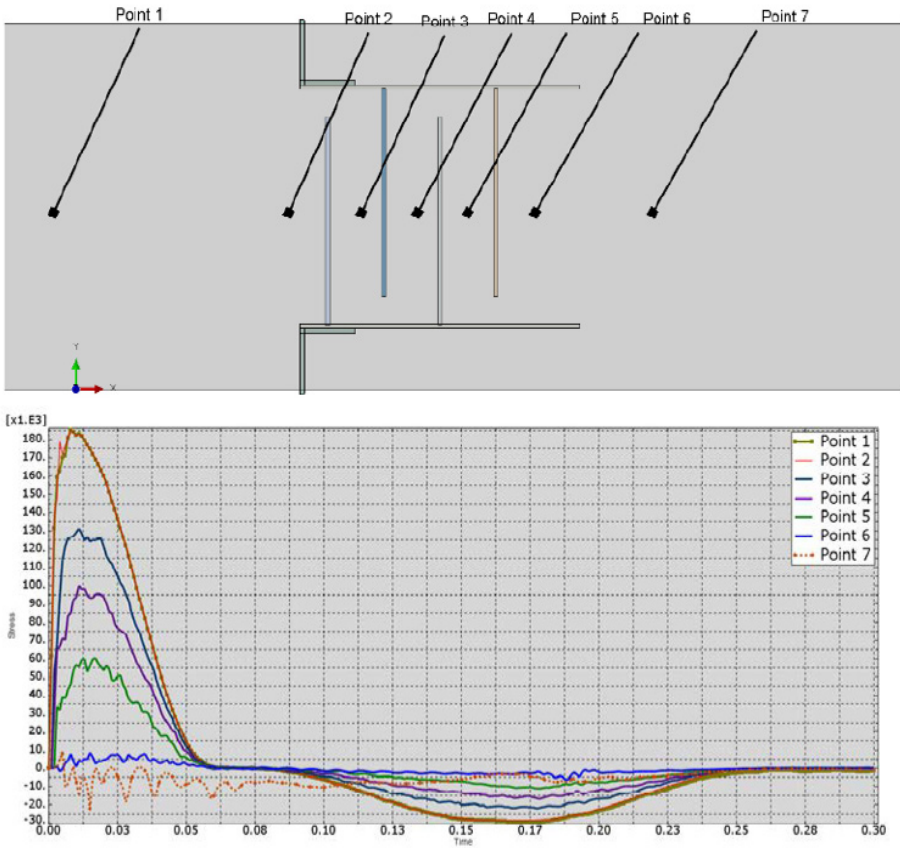


Figure 6. Overpressure distribution in various points in damper.

Source: SIMULIA Community Conference, 2014

Figure 7 show the values of minimum and maximum pressure measured in damper points. Achieved values show that over pressure gradually decreased.

Figure 7. Overpressure in selected points.

	1 (KPa)	2 (KPa)	3 (KPa)	4 (KPa)	5 (KPa)	6 (KPa)	7 (KPa)
$\Delta P+$	185	185	132	101	60	8	5
$\Delta P-$	-29	-29	-22	-17	-11	-6	-19

Stress analysis of damper performed. Plastic deformation point shown on Figure 9. Maximum stress achieved in model is 250 MPa, which lead to yielding of St 3 material. Yielding point located in channel plate attachment point.

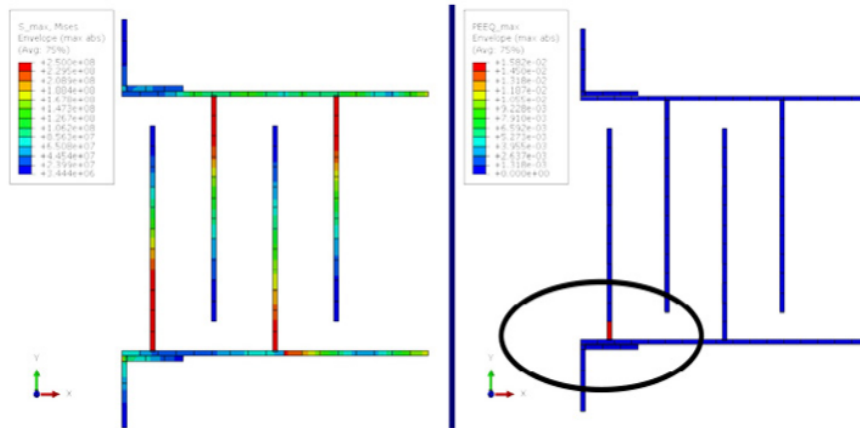


Figure 9. Stress and equivalent plastic strains distribution in damper.

Pressure reduction coefficient k_{loc} calculated for damper. Coefficient determined as $k_{loc} = P_{in}/P_{out}$, where P_{in} – pressure before extinguisher; P_{out} – pressure after extinguisher. Pressure reduction coefficient for analyzed extinguisher design is 18.

Achieved results let us made a hypothesis that safety devise is impenetrable for blast wave with selected parameters, and consider protective devise as continuous wall. This assumption allow us considerably decrease size of safety devise blast wave interaction simulation. For further steps safety device considered as rigid – stress analysis of full devise for acting pressure will made after the full analysis.

To determine the overpressure level acting on operation station structure, simulation model of blast wave interaction with building made. Sizes of the building is following: height – 4 meters, length - 15 meters. Explosion occurs in 65 meters away from building front wall. Blast wave interaction with building shown on Figure 10.

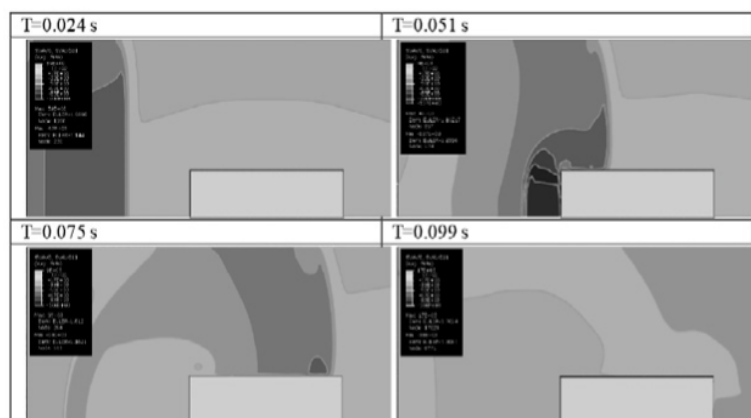


Figure 10. Blast wave interaction with unprotected building.

Source: SIMULIA Community Conference, 2014

Pressure distribution over the building divided to specific areas: Flow separation area – areas near the edges of building walls; wall central area – area in the center of wall where pressure distribution is regular. Overpressure values for considered areas was achieved in simulation, and provided on Figure 11

	Pressure
Maximum of building	178 kPa
Front wall	125 kPa
Roof	62 kPa
Back wall	36 kPa

Figure 11. Overpressure on specified building areas

Due to regulation [2] overpressure higher than 100 kPa lead to complete destruction of buildings. Hence, it is necessary to decrease overpressure on operational station wall.

Simulation model with building covered by safety device made. Safety device with following parameters is analyzed:

- Safety device height – 4 m., 5.5 m., 7 m., 8.5 m.
- Device location distance from building front wall - 2 m., 4 m., 6 m., 8 m., 10 m., 12 m.

Results of simulation organized in diagrams, which show pressure-height-distance dependence. Achieved diagrams allow to preliminary design the safety device parameters to protect the building with similar sizes from blast load with similar parameters. Global pressure reduction coefficient (GPRC) diagram shown on Figure 12. Pressure diagrams for different areas shown of Figures 13 and 14.

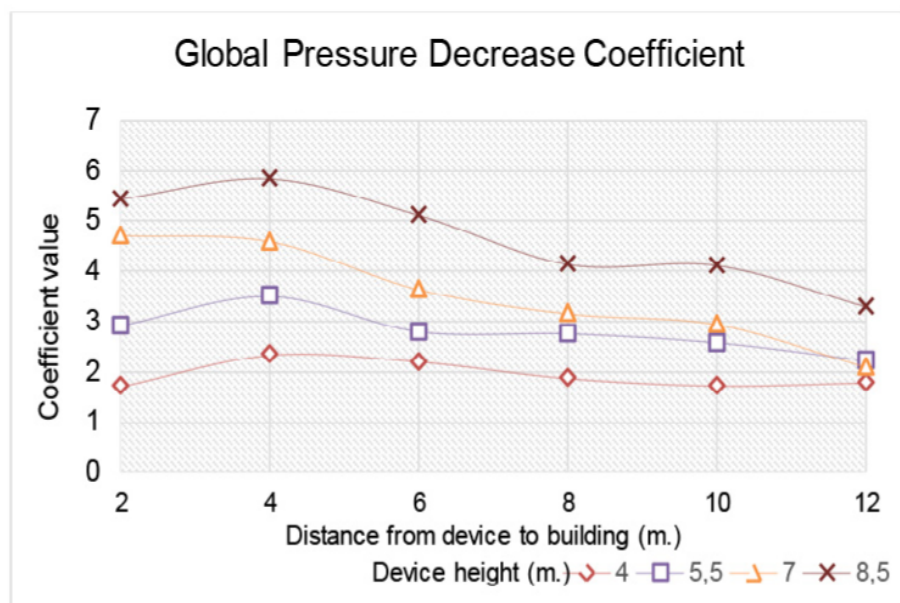


Figure 12. GPRC dependence from device height and distance from protected structure.

Source: SIMULIA Community Conference, 2014

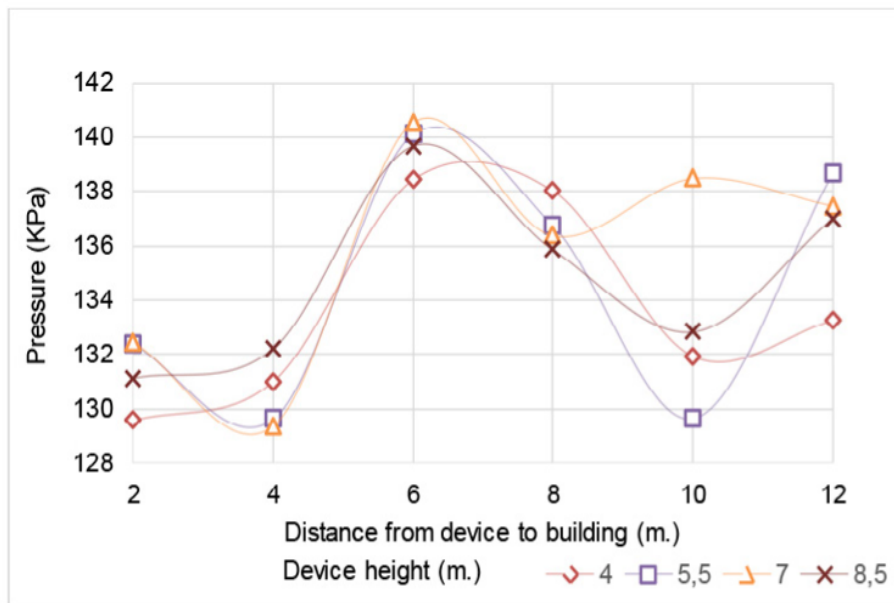


Figure 13. Pressure on device front wall depended from device height and distance from protected structure.

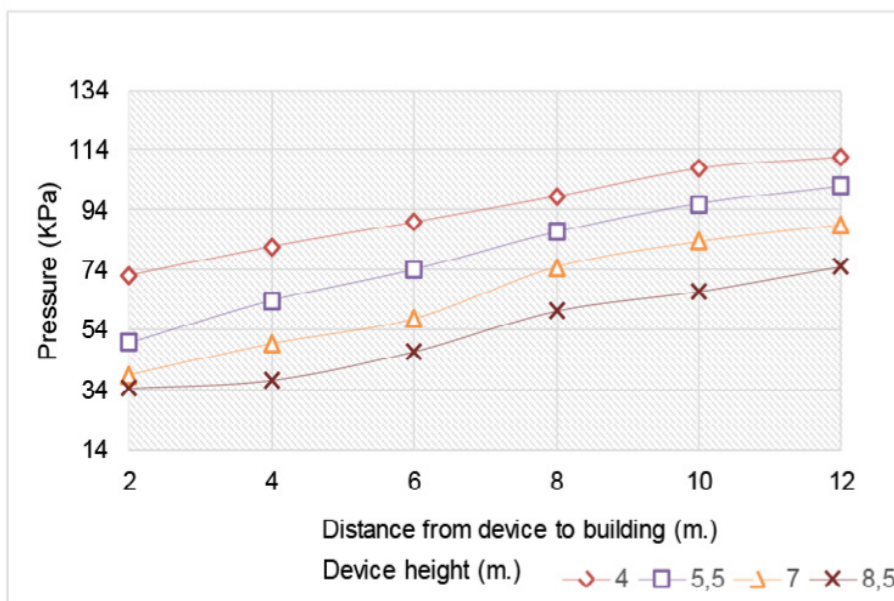


Figure 14. Pressure on building front wall dependence from device height and distance from protected structure.

5. MODEL AND ANALYSIS TIME

HP EliteBook 8570w with Core i7-3840QM (4x2.80 GHz cores), 32 GB RAM used to run analysis.

Three types of model was used do made provided analysis.

1. 3-dimensional CEL model of dampener
2. Quasi 2-dimensional CEL model of dampener
3. Quasi 2-dimensional CEL model of protective devise and dampener

Source: SIMULIA Community Conference, 2014

Run of model 1 take a ~12 Hours on mentioned PC. Model 2 created to decrease analysis run time. Run of model 2 take a ~1 hour on mentioned PC. Results from model 1 & 2 compared, and model 1 is more conservatives, than model 2.

6. ANALYSIS VERIFICATION

To verify CEL method adequacy for blast wave – structure interaction analysis we created special model to verify simulation results and real life test data.

Comparison of simulation and experimental data of blast wave interaction with brick wall made and published in paper [4]. Achieved results shows difference between simulation and experiment is no more than 3-7%.

7. CONCLUSION

Achieved results confirm that safety device can decrease overpressure due to blast wave load on protected structures. This allows putting the device on existing oil & gas plants, instead of rebuilding of existing infrastructure.

Provided method allows designing and analyzing safety device for different blast wave parameters. Coupled Euler-Lagrange method allow to model simulation of blast wave interaction with damper. It is possible to use method for approximate analysis of blast loads acting on protective device.

Future research will consider solution of full 3-dimentional analysis of blast wave interaction with protective devise frame with installed dampeners.

8. REFERENCES

1. Tlyasheva R.R., Smolnikov O.L., Tropkin S.N., Bayazitov M.I., "Obespechenie zashishennosti operatornoi neftepererabativaushogo predpriyatiya ot vozdeistviya udarnoi volni", Mirovye soobshchestvo: Problemy i resheniya, T.30 pp 71-73, Ufa, 2011.
2. RD 03-409-01 Metodika otsenki posledstviy avariynih vzrivov toplivo-vozdushnih smesey. Utverzhdena postanovleniyem Rostekhnadzora Rossii 26.06.01
3. PB-09-540-03 Obshchie pravila vzrivobezopasnosti dlya vzryvopozharoopasnykh hemicheskikh, neftehimicheskikh i neftepererabativaushich proizvodstv. Moscow, GUP "NTC "Promyshlennaya bezopasnost", 2003.
4. S.N. Tropkin, R.R. Tlyasheva, M.I. Bayazitov, Z.R. Rafikova, I.R. Kuzeev "Verification of a mathematical model of a shock wave in An emergency at the plant of oil grown", Oil and Gas business. Electronic Scientific Journal. 2013 No.1. pp 476-486.

Source: SIMULIA Community Conference, 2014

Pipeline Rupture in Abaqus/Standard with Ductile Failure Initiation

SUMMARY

Defects may be introduced in metal pipelines during construction, repair, or by accident. A common example is a gouge from a backhoe bucket or other heavy equipment. At the site of a local defect operating stresses may become sufficiently concentrated to induce plastic deformation and material damage, possibly resulting in eventual failure of the pipeline.

Historically, methods for assessing the structural integrity of a damaged pipe have been based on experimental tests. The Abaqus finite element suite includes the ability to simulate the initiation and evolution of damage in metals, providing a low-cost alternative to laboratory structural testing. In this Technology Brief, Abaqus/Standard will be used to predict the burst pressure of a steel pipeline with a notch-type defect. A ductile damage initiation criterion is used, and favorable comparison with available experimental data will be shown.

BACKGROUND

Pipelines are a critical component of industrial infrastructure and are used world-wide to transport liquids and gases. During the course of its lifetime, a metal pipeline may sustain mechanical damage in such forms as dents, gouges, or weld defects. When damage is detected, a decision to monitor, repair, or replace is necessary.

A body of assessment guidelines for determining the fitness-for-purpose of a damaged pipeline has been built over the past several decades. As discussed in [1], many of these methods rely on experimental results and semi-empirical procedures; as

such, their validity may be limited when considering loadings, materials, or specific damage configurations that are outside the scope of their assumptions.

With the ability to include the effects of damage initiation and evolution in the analysis of a ductile metal pipeline, the Abaqus finite element suite can complement existing methodologies by adding a more general predictive capability. Specifically, two types of damage initiation criterion are offered: ductile, based on the nucleation, growth, and coalescence of voids, and shear, based on shear band localization. We will focus on the use of the ductile criterion. The present analysis will consider an internally pressurized pipe of API X65 steel with a gouge defect.

FINITE ELEMENT ANALYSIS APPROACH

The geometry of the model under consideration is shown in Figure 1. A simulated gouge, 100mm long, was introduced into the pipeline. A quarter-symmetric mesh of second order hex elements was generated, and internal pressure loading was applied. End forces were applied to simulate experimental closed end conditions, and the loads were increased linearly with time. In general, the specification of damage initiation is included in the material definition and must be used in conjunction with a plasticity model. In this analysis we use the Mises plasticity formulation. The mesh is shown in Figure 2.

Key Abaqus Features and Benefits

- Damage initiation and failure modeling for ductile metals
 - Ductile and shear initiation criteria allow for the modeling of two primary fracture mechanisms: coalescence of voids and shear banding
 - Can be used for bulk or sheet metal analyses
 - Available for all mechanical elements and allows for element removal

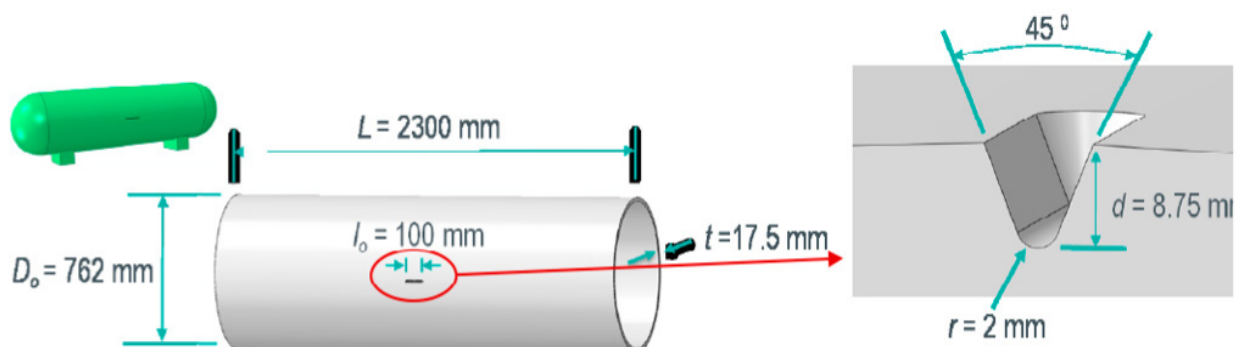


Figure 1: Geometry of damaged pipe model, with detail of simulated gouge

Source: Abaqus Technology Brief

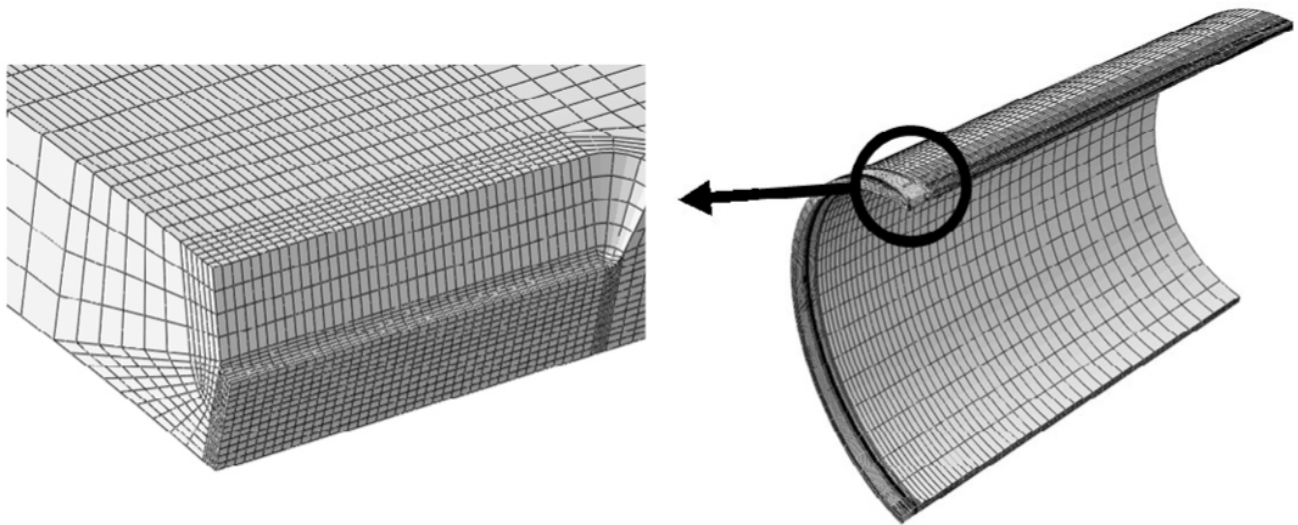


Figure 2: Quarter symmetric mesh of the pipe, with close-up of detailed mesh at the gouge

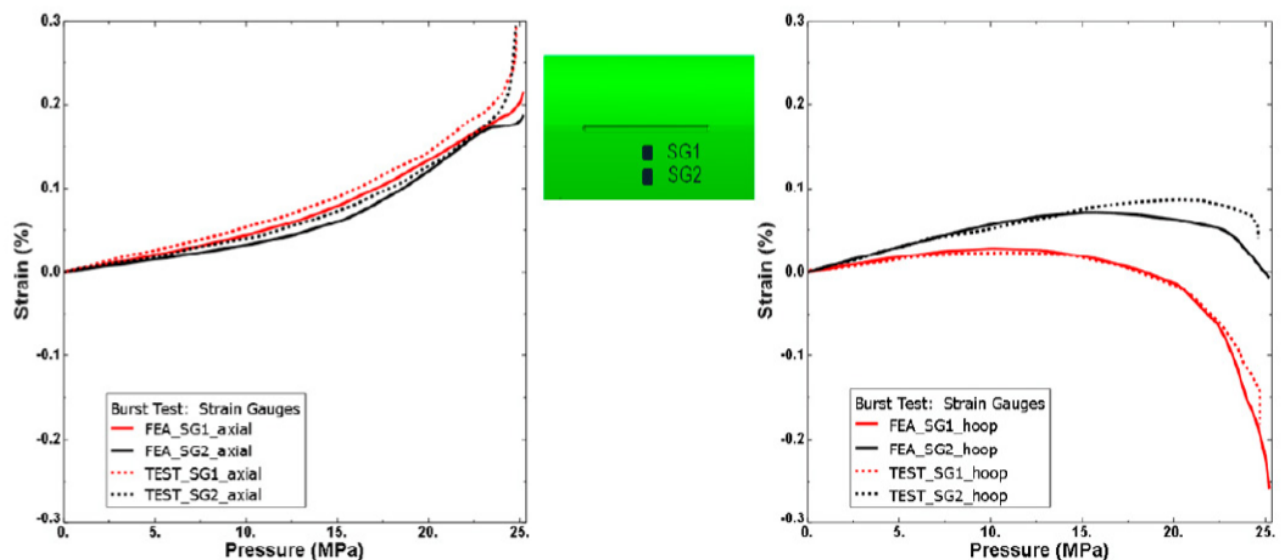


Figure 3: Experimental and analytical strain results near the notch and location of strain gauges

The ductile damage initiation criterion is a phenomenological model. It is included in the analysis by specifying the equivalent plastic strain at damage initiation as a function of stress triaxiality and equivalent plastic strain rate. Stress triaxiality is defined as $h = -p/q$ (where p is the pressure stress and q is the Mises equivalent stress), and is known to play a role in damage growth. The procedure for calibrating the model used here is outlined in Appendix A.

RESULTS

We compare the Abaqus/Standard results to the full scale experimental burst test data collected by Oh et al [2]. Axial

and hoop strain measurements from two gauges are plotted against the analytical result in Figure 3. The gauges are located adjacent to the gouge, at circumferential distances of 24.9 and 54.9 mm. Favorable comparison with the experimental data is achieved.

A contour plot of the damage initiation output variable DUCTCRT is shown in Figure 4. Damage has initiated when this variable is greater than 1.0. From the contour, we see that the critical element in the structure is in the root of the notch at the intersection of the symmetry planes. By X-Y plotting the initiation criterion in the critical element, a more precise determination of the failure pressure can be determined. In

Source: Abaqus Technology Brief

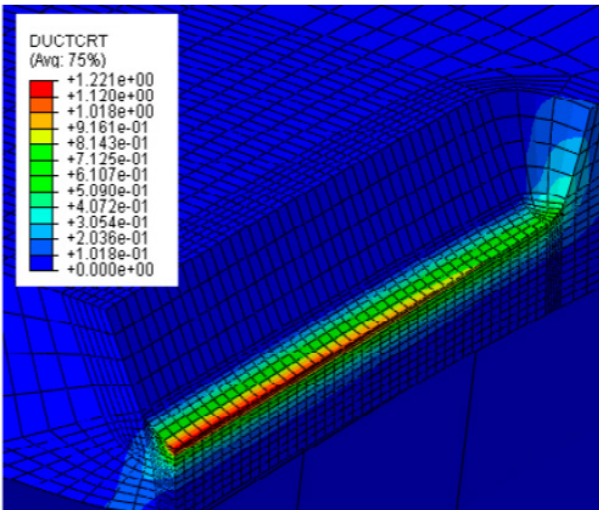


Figure 4: Damage initiation criterion in the pipe gouge

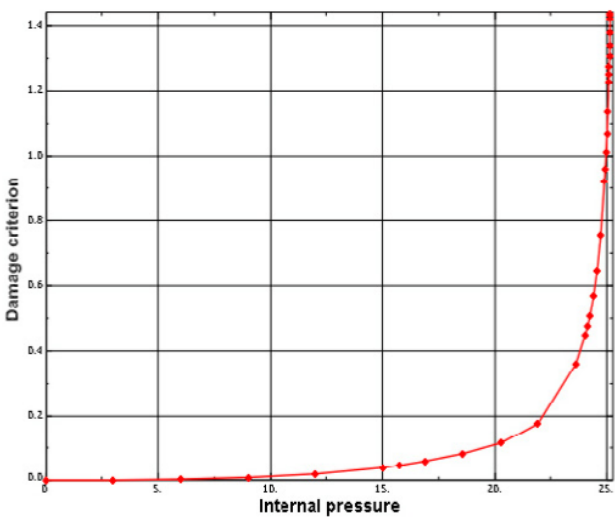


Figure 5: Damage criterion v. internal pressure load in critical element

Table 1: Comparison of failure criteria

Criterion	Burst Pressure Predicted/ Experimental
Abaqus Ductile Failure	1.01
Net Section Collapse	1.04
Peak Max Principal Stress	1.01
Peak Equivalent Plastic Strain	1.01
Axial Crack (ASME B&PV Code Sec. XI)	0.86

Figure 5, DUCTCRT at the centroid of the critical element is plotted against the applied internal pressure, and the threshold of 1.0 is crossed at a pressure of 24.97 MPa. The experimentally determined burst pressure 24.68 MPa.

The Abaqus prediction is compared to several other commonly used failure criteria in Table 1. The relatively good performance of the peak criteria is attributed to the same relative triaxiality of the smooth round tensile bar (~0.65) used to determine material properties as compared to that of the actual application (~0.6).

CONCLUSIONS

In this Technology Brief we have demonstrated the utility of the Abaqus/Standard ductile failure initiation criterion for predicting the burst pressure of pipes with notch-type defects. Good correlation with available full scale experimental data has been shown. The Abaqus damage initiation and evolution capability for metals provides a general numerical tool that can supplement existing failure prediction methods that are based on empirical data.

REFERENCES

- Cosham, A. and Hopkins, P., "An overview of the pipeline defect assessment manual (PDAM)," 4th International Pipeline Technology Conference, 2004.
- Oh, C.-K., Kim, Y.-J., Baek, J.-H., Kim, Y.-P., and Kim, W.-S., "Ductile failure analysis of API X65 pipes with notch-type defects using a local fracture criterion," International Journal of Pressure Vessels and Piping, Vol. 84, pp. 512-525.

Abaqus References

For additional information on the Abaqus capabilities referred to in this document please see the following Abaqus 6.13 documentation references:

Analysis User's Guide-"Damage and failure for ductile metals," Section 24.2.1

Source: Abaqus Technology Brief

Appendix A - Determination of the Ductile Failure Initiation Parameters

To use the ductile failure initiation criterion in Abaqus/Standard, one must specify the equivalent plastic strain at damage initiation as a function of stress triaxiality and strain rate. As outlined in [2] this requires an experimental program, a finite element analysis of each test, and the construction of a failure loci.

Experimental tests were performed on round, notched bars. The notched specimens were used to capture the effect of stress triaxiality on yield and tensile strength. A schematic diagram of the bar geometry is shown in Figure A1. The bars had an outer diameter of 17.5 mm and length of 130 mm. Smooth bars and those with three different notch radii were tested: 1.5, 3 and 6 mm. The bars were loaded in tension until complete fracture was achieved.



Figure A1: Schematic geometry of round, notched tensile test specimen

Each test had a corresponding axisymmetric finite element analysis. Second order, reduced integration elements (CAX8R) were used in a half-symmetric mesh. In the critical location, an element size of 0.15 mm x 0.15 mm was used. The Mises plasticity model was employed, and nonlinear geometric effects were included. A comparison of the computed stress-strain response and the experimental measurements is shown in Figure A2. The Abaqus/Standard analyses were run until crack initiation occurred in the experiment and good correlation was obtained. It can be seen that as notch radius decreased, yield and tensile strengths increased, but strain to failure decreased. This behavior is consistent with the increasing triaxiality of the stress state with decreasing notch radius.

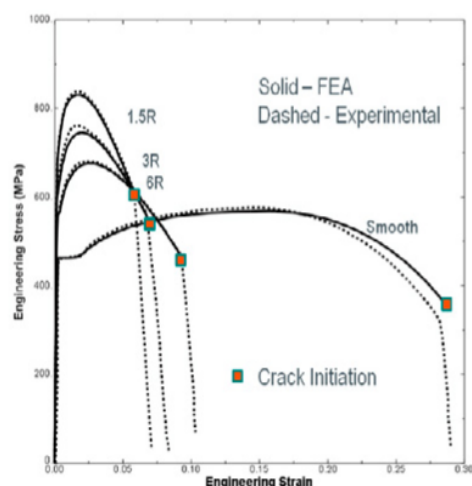


Figure A2: Experimental and Abaqus results for smooth and notched bar tensile tests

The critical location for each of the test specimens is at the center point of the bar. From the analysis results, the equivalent plastic strain as a function of the stress triaxiality at that location is plotted in Figure A3.

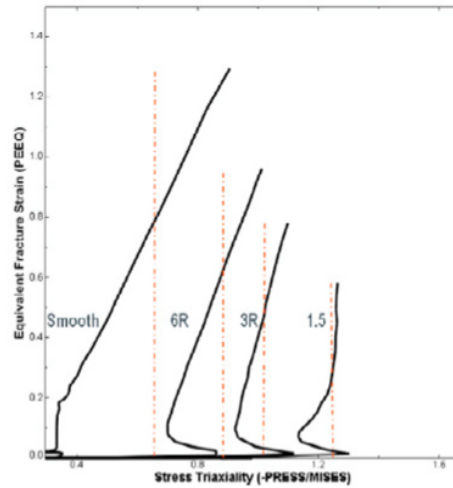


Figure A3: Equivalent strain vs. stress triaxiality, with values of average stress triaxiality

Included in Figure A3, shown by red dotted lines, is the average stress triaxiality for each specimen, defined as

$$\left(\frac{\sigma_{press}}{\sigma_{equiv}} \right)_{avg} = \frac{1}{\epsilon_{ef}} \int_0^{\epsilon_{ef}} \frac{\sigma_{press}}{\sigma_{equiv}} d\epsilon_{equiv}$$

where ϵ_{ef} is the equivalent strain at failure initiation. Each equivalent strain trace in Figure A3 ends at its corresponding value of ϵ_{ef} . The points located by the black squares in Figure A4 are the equivalent strain—stress triaxiality data points used in the Abaqus ductile failure initiation definition. The red curve is the loci fitted by Oh et al. [2].

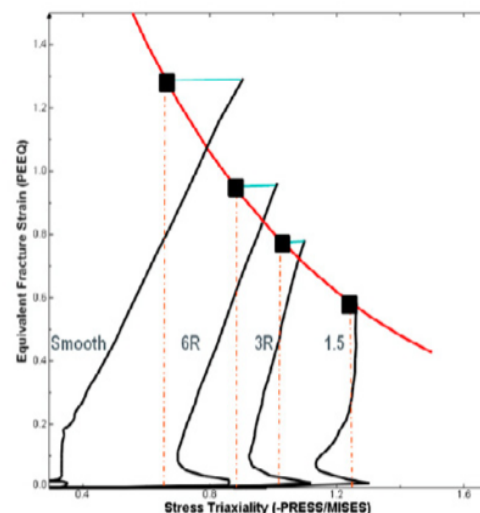


Figure A4: Equivalent strain—stress triaxiality pairings (black squares) used in the Abaqus ductile failure initiation criterion

Source: Abaqus Technology Brief

Our **3DEXPERIENCE®** platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3DEXPERIENCE®** Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 190,000 customers of all sizes in all industries in more than 140 countries. For more information, visit www.3ds.com.



3DEXPERIENCE®



| The **3DEXPERIENCE®** Company

Americas

Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

Europe/Middle East/Africa

Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific

Dassault Systèmes K.K.
ThinkPark Tower
2-1-1 Osaki, Shinagawa-ku,
Tokyo 141-6020
Japan