Proceedings of PVP2006
2006 ASME Pressure Vessels and Piping Division Conference
July 23-26, 2006, Vancouver, British Columbia, Canada

PVP2006-ICPVT11-93632

Analysis of Pressure Vessel Sloshing

S.M. McGuffie
Porter McGuffie, Inc.
Lawrence, KS, USA
Phone: 785.856.7575
Email: sean@pm-engr.com

M.A. Porter
Dynamic Analysis
Lawrence, KS, USA
Phone: 785.843.3558
Email: mike@dynamicanalysis.com

ABSTRACT

ASME BPVC Section VIII Division 1 Paragraph UG-22 (f) requires consideration of the loadings from seismic conditions. For a vessel containing a fluid, the loading due to sloshing must be considered. ASCE Standard 7-02 (Section 9.14.7.3) states that a damping value of 0.5% can be used to account for the fluid sloshing. This can lead to an overly conservative design by over-estimating the loads on the tank structure. A time-history analysis was performed on a horizontally mounted pressure vessel experiencing 3-axis time history loads in order to determine if this method is more accurate in determining the loads. The analysis employed a 3-dimensional computational fluid dynamics (CFD) model, using transient time-history techniques. The reactions at the mounting locations were compared to the reactions computed using closed form solutions, demonstrating good correlation. The results show that CFD is an excellent tool for investigating seismic sloshing loads in vessels.

INTRODUCTION

Fluid sloshing is of concern in pressure vessels that may be subjected to seismic events. Traditional methods of analysis have involved the use of modal analysis techniques [1,2].

The disadvantage of using modal techniques is that very little time-history data can be extracted regarding the forces placed on the structure. Additionally, these techniques assume that the fluid's surface stays nearly horizontal during the duration of the event. It has been the authors' experience that during the analysis of some seismic events this assumption becomes invalid as the fluid can slosh throughout the volume of the tank imposing occasional uplift loads on the top of the structure due to fluid pressures.

This paper explores the use of computational fluid dynamics (CFD) models for the investigation of sloshing loads occurring on a generic pressure vessel. These results are then compared to empirical results to determine the applicability of CFD for the analysis of pressure vessel sloshing.

BACKGROUND ON COMPUTATIONAL METHODS

To this point the use of computational methods has been limited for the analysis of sloshing in pressure vessels. The primary reason for this has been the imposing run times and costs associated with performing the analysis. There are several reasons why the run times for Volume of Fluid (VOF) problems have historically been unmanageable in an engineering environment, including:

- The analysis of a sloshing problem requires the use of the VOF model to track the interface between two immiscible fluids. Because the shape of the interface is changing over time, this implicitly makes the analysis transient. Additionally, due to the nature of the VOF model, a small enough time-step must be used so that the interface can be effectively tracked. Since earthquakes are 20-30 second duration events, this can result in the need to analyze millions of time-steps.

- Industrial pressure vessels typically enclose very large volumes while a fine enough mesh density must be used to capture the interface and the flow parameters. This leads to a very large number of cells being required for the accurate solution of a problem.

The increases in chip speeds and the ability to solve CFD problems on parallel networked computers has greatly decreased the time required to perform long duration transient analyses on large models. It is envisioned that these classes of problems will become routine in the near term.

DESCRIPTION OF PROBLEM

The problem considered during this analysis was a 3.658m diameter, 12.192m long horizontally mounted pressure vessel with hemispherical heads. This vessel was subjected to a time history acceleration in three axes taken from the Loma Prieta earthquake. Due to time constraints in performing the analysis and the small time-step that was required, only 10 seconds of the earthquake was considered for this analysis. The time-history acceleration data the model was subjected to is shown in Figure 1.

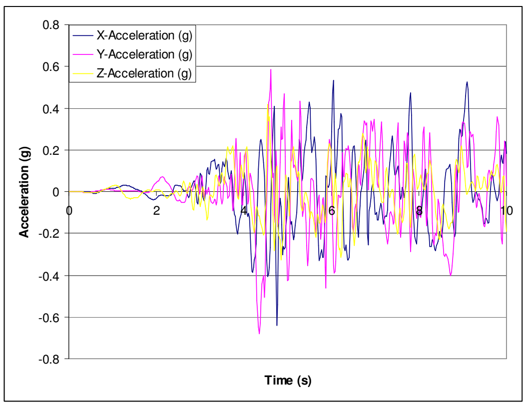

Figure 1 – Time history acceleration considered for the problem

The tank was assumed to be half full with working fluids of water and air. The material properties used for the analysis are contained in Table 1.

Material	Density (kg)	Viscosity (kg/m-s)
Water	998.2	1.0030E-03
Air	1.225	1.7894E-05

Table 1 – Material properties used for the analysis

EMPIRICAL SOLUTION

Platyrrachos et. al. [1] provide a solution for the fundamental sloshing frequencies and associated forces using a finite element method. In this paper they find that the normalized frequency (λ_n) for the first mode is 1.3557 sec^4, as shown in their Table 1.

Using their definition of λ_n as defined in Table 1 of their report:

$$\lambda_n = \varpi_n^2 \, R/g \qquad (1)$$

and substituting the values from this analysis:

λ_n = 1.3557 sec^4
R = 1.8288 m
g = 9.81 m/s^2

it can be determined that the fundamental sloshing frequency for the tank as predicted through empirical methods is 0.429 Hz.

Figure 6 of their report shows the variation of sloshing (convective) and impulsive mass ratios. From this figure for a tank with a liquid height parameter of 0, it can be found that M_{1c}/M_L = 0.59 (Convective ratio) and that M_I/M_L = 0.4 (Impulsive ratio).

COMPUTATIONAL SOLUTION

A CFD model of the vessel was constructed. This model was then analyzed using VOF techniques within Fluent 6.0.

DESCRIPTION OF MODEL

The model was constructed by revolving a 2-dimensional profile of the vessel along its centerline. The resulting model consisted of 119,340 six and eight node elements with 119,977 nodes. Figure 2 contains an image of the grid used for the analysis.

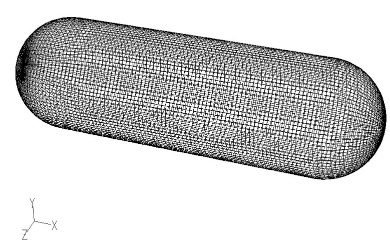

Figure 2 – Computational grid used for analyses

SOLUTION TECHNIQUES

The model was solved in Fluent 6.0 using a transient analysis methodology. To perform the analysis a journal file was created. This file modified the applied gravitational accelerations to the model based on the earthquake accelerations at the modeled time and then iterated a given number of time-steps. Model data files were written every 0.02 sec of solution time. This corresponds to the sampling frequency of the Loma Prieta earthquake data. Additionally, the following options were selected:

- The Renormalization of Groups (RNG) k-ε turbulence model was used with Standard Wall Functions. This is the simplest "complete model" of turbulence with the RNG correction allowing for low Reynold's number turbulence to be included [3]. The standard values for the turbulent constants were used. This model was chosen as it is a good general purpose model with a small solution overhead.

- The VOF model was used to track the interface between the two fluids. This model solves a single set of momentum equations for the entire domain and tracks the interface between the fluids with four different options [4] for tracking the interface. For the results presented in this report, the Specified Operating Density option was not enabled.

- The model was analyzed with a time-step of $5 * 10^{-4}$ seconds. This time-step was chosen because it was found, through experimentation, to be the maximum stable time-step for the VOF surface tracking algorithm. Due to the limitations of the VOF model, only First Order Implicit Time-Stepping Schemes were employed.

COMPUTATIONAL RESULTS

An animation was produced of the entire event. This allowed for visualization of the wave patterns within the vessel and also provided a visual check of the forces that were provided. Figure 3 shows one frame of the animation (t=7 sec).

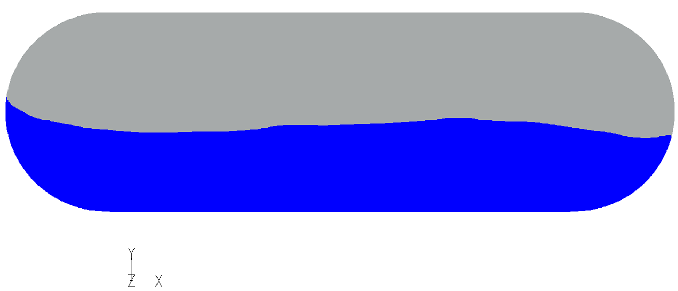

Figure 3 – Wave motion (represented by blue area) for time = 7 sec

Before beginning the analysis, a check was performed on Fluent's reported weight. Fluent's reported model weight was 568,456.58 N which compares favorably to the actual weight of 563,854.31 N, <1% difference.

The force data in all three axes was queried from the analysis results. Due to Fluent's limitation of only being able to report forces in one direction to an output file during solution, this querying had to be performed manually. The authors chose a sampling time scale of 0.06 sec, although more data was available. Figures 4, 5 and 6 show the time-history force results on the vessel, in the X, Y and Z directions respectively, for the peak forces that were recorded.

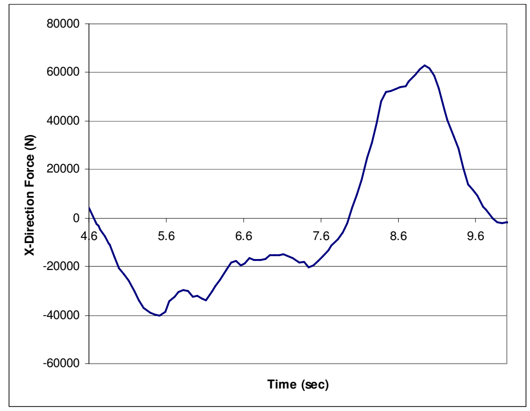

Figure 4 – X-direction forces due to ground excitation

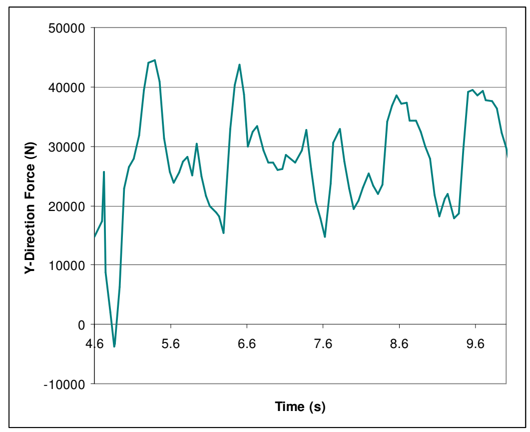

Figure 5 – Y-direction forces due to ground excitation

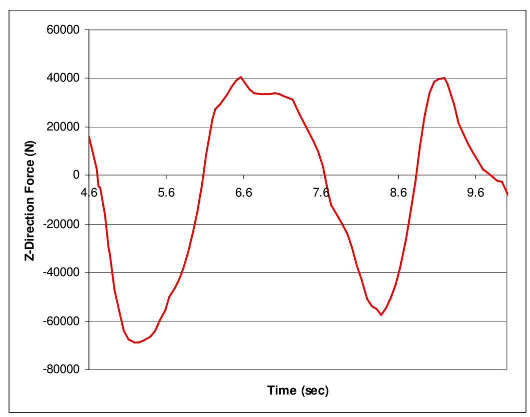

Figure 6 – Z-direction forces due to ground excitation

As can be seen from the results presented in Figures 4 and 6, the lateral and longitudinal loads show a variability of approximately 100,000 N peak-to-peak, while the Y-direction force variance was approximately 45,000 N peak-to-peak. It should be noted that the Y-direction force graph does not contain the component due to gravity (568,456.58 N). Table 2 contains the maximum and minimum forces calculated from the analysis.

Direction	Maximum Force (N)	Minimum Force (N)
X	62872	-40129
Y	44484.2	-3246.6
Z	40488.3	-68813.9

Table 2 – Maximum and minimum force values from the analysis

Additionally, a Fast Fourier Transform (FFT) was performed on the output data in all three directions to determine the primary modes of sloshing. Figure 7 shows the results of this analysis.

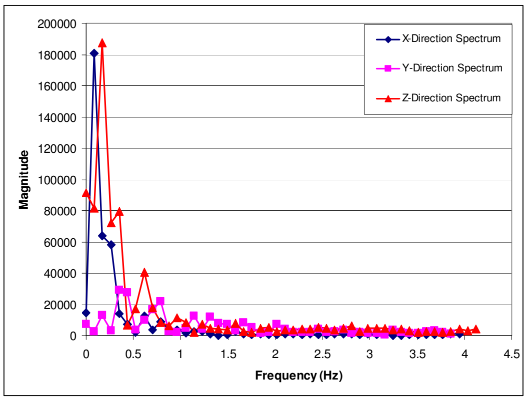

Figure 7 – FFT spectrum results from the analysis

As can be seen from Figure 7, the Z-direction has the highest magnitude from the FFT analysis. The Y-direction displays the lowest magnitude. Table 3 lists the frequencies for the peaks in each direction.

Direction	Peak Frequency (Hz)
X	0.088
Y	0.789
Z	0.175

Table 3 – Frequencies at which peaks occur from FFT analysis

To compare the force values computed by CFD to those computed through empirical methods, the Convective and Impulsive Masses (as described in the Empirical Solution Section of this report), were calculated for the Fluent model. To perform this computation, the Impulsive force was defined as the product of the gravitational acceleration and the fluid mass. The difference between Fluent's reported force and the Impulsive force was the Convective force. These force values could then be converted into equivalent masses and the ratio of the masses could be determined. Table 4 contains this data.

	X - Direction	Y - Direction	Z - Direction
Fluent Force (N)	4145.32	-13736.86	16026.92
Gravitational Accel. (g)	0.27	-0.49	0.23
Impulsive Force (N)	-15366.84	28404.12	-13260.12
Impulsive Mass (kg)	1566.45	2895.42	1351.69
Convective Force (N)	19512.16	-42140.98	29287.04
Convective Mass (kg)	1989.01	4295.72	2985.43
Convective Force %	0.56	0.60	0.69

Table 4 – Comparison of convective and impulsive masses

COMPARISON OF EMPIRICAL AND COMPUTATIONAL RESULTS

As can be seen from the results presented in Figure 7 and Table 3, there is very little agreement between the predicted first sloshing frequencies from the CFD model and those from the empirical methods described in Reference 1. The authors believe there are two possible reasons for this:

- Inclusion of fluid viscosity in the CFD model that will lead to damping of the fluid system and a reduced sloshing frequency.

- The use of 3-D time-history acceleration rather than modal techniques to compute the response has affected the system's response.

As can be seen in Table 4, the convective mass ratios compare very favorably to those computed in the Platyrrachos paper.

DISCUSSION OF APPLICATIONS TO INDUSTRY

As parallel computing resources become more readily available, the use of CFD to study the sloshing impacts of seismic loads on pressure vessels will become more feasible within the constraints of an engineering environment. Discussions are given below regarding some of the areas where computational methods will likely give better results than standard empirical models.

NON-STANDARD GEOMETRIES

While closed-form solutions exist for many standard pressure vessel geometries, sometimes the closed-form solution may not be representative of the actual vessel geometry. Such cases may occur due to internal components or baffling that obstruct the flow path considerably. In the past the impact of baffles or internal components has traditionally been determined through the use of scale models where numerical techniques can now be used to study a component's impact.

COUPLING TO FEA MODELS

Using user-developed routines and/or routines developed by software companies, the pressure distributions determined during a CFD analysis can be directly passed to a finite element (FE) model for performing a transient FE analysis. This can be critical in the evaluation of components and the vessel itself where the time-history pressure distribution affects the response.

FUTURE TRENDS

The advent of inexpensive and fast computing power has allowed for more situations to be analyzed using computational methods rather than through typical empirical techniques. This trend will continue as chips become faster and parallel computing power becomes less expensive. Additionally, the migration to 64-bit computing architecture allows for a much larger model to be considered using a single processor (past limits were ~1 million cells per processor). This ability to perform analyses relatively quickly on large models will allow for the analysis of even more situations.

CONCLUSION

A transient CFD analysis has been conducted to study the response of a generic pressure vessel subject to 3-axis ground acceleration.

It has been found that the fundamental modes predicted by the CFD analysis are much lower than those predicted by empirical methods. Several reasons have been given on why this discrepancy may occur. These include the inclusion of viscosity in the CFD model and the use of a 3-D time-history analysis rather than modal techniques to predict the structure's response.

Very good agreement has been found between the Convective Mass predicted by Fluent and that predicted by Platyrrachos et. al. Since Fluent's original mass for the vessel was within 1% of the actual vessel mass and this level of agreement has been reached, confidence can be placed in the results computed by CFD.

FUTURE WORK

It is likely that CFD analysis of pressure vessel sloshing will become common in the near future. Therefore, to develop the proper procedures for such analyses the authors plan to continue work on modeling this phenomenon. The purpose of this work will be to determine the effects of more solution choices on the system's response. Specific choices to be studied include:

- Effects from the use of various turbulence models
- Effects of mesh density on the calculated results
- Effect of the use of the Specified Operating Density parameter for the VOF model

Work has already begun on determining the influence of some of these parameters on the solution.

REFERENCES

1. Platyrrachos, M. A. and Karamanos, S. A. (2005), "Finite Element Analysis of Sloshing in Horizontal-Cylindrical Industrial Vessels Under Earthquake Loading", *Proceedings of PVP2005*
2. Housner, G. W. (1957), "Dynamic Pressures on Accelerated Fluid Containers", *Bulletin Seismological Society of America*, Vol. 47 pp 15-35
3. Fluent 6.0 Manual, User's Guide Section 10.4.2
4. Fluent 6.0 Manual, User's Guide Section 20.1.1