Proceedings of -PVP 2007
2007 ASME Pressure Vessels and Piping Division Conference
July 22-26, 2007, San Antonio, Texas, USA

PVP2007-26009

STUDY OF DYNAMIC STRESSES IN PIPE NETWORKS AND PRESSURE VESSELS USING FLUID-SOLID-INTERACTION MODELS

Philip Diwakar
Bechtel National Inc.
3000 Post Oak Blvd
Houston, TX, 77056 USA
Email: pmdiwaka@bechtel.com

Lorraine Lin
Bechtel National Inc.
50, Beale Street
San Francisco, CA, 94105 USA
Email: lhlin@bechtel.com

ABSTRACT

A lack of understanding the fluid-structure interactions has resulted in a number of infamous structural failures in the past. For example, the collapse of the Tay Bridge in Scotland in 1879, of the Tacoma Bridge in 1948 and of three tall cooling towers in Ferrybridge/England in 1965 have been intrinsically related to fluid forces acting on the structure. Flutter, flow induced vibration, divergence and related phenomena may be studied using the Fluid-Solid-Interaction (FSI) approach.

This paper gives three examples of the FSI approach and shows the innovative application of state-of-the-art computational methods to improve realism and accuracy in engineering analyses.

Case 1: Study of Hydrodynamic Sloshing Loads: The sloshing of liquid in large vessels under seismic loads is an active topic. The movement of the free surface of the liquid is simulated using a two-phase volume of fluid model at various liquid heights. The transient forces generated by the fluid on the vessel wall and internals are superimposed as loads on a dynamic non-linear calculation and the fatigue and stresses are computed in an explicit finite element analysis. This approach calculates the local sloshing effects on internals as opposed to the traditional approach of using spring-mass elements.

Case 2 – Bending of Large Pipes due to Temperature Differentials: Pipe temperature differentials can be caused by either extremely cold liquids or hot liquids flowing at the bottom of a piping system while the top is exposed to atmospheric conditions. Differential expansion can cause pipe deformation resulting in pipe lift-off at its supports and failure at the weld locations and T-joints. Heat transfer from complex multi-phase flows was simulated using CFD. The predicted pipe wall temperatures were then input to an FEA grid and analyzed for heat transfer and thermal stresses. These stresses were compared to ASME-standard allowable limits. Based on this analytical approach, a design guide for various diameters of flare header pipes, supports and tees has been established. Details of this paper were previously published in [Ref 1] and are not described in detail in this paper.

Case 3: Establishing velocity limits and line sizing criteria in pipes: The original guidelines in Fluid Flow Manuals were developed over the last fifty years based on project experience and economic and best practices technology of the time. The criteria have proven out as good, but overly conservative with regards to line size. Compressor discharge guidelines are based on the erosion velocity limits. Based on a dynamic analysis approach – using unsteady flow rates from compressors - stresses due to flow induced vibration, noise and fatigue, hydraulic transients such as water hammer effects for long lines (greater than 1000 feet), flashing and control valve cavitations may be studied. FSI was used to determine if the velocity limit guidelines hold in the current designs and use a parametric approach to mitigate the bottlenecking by supplying a simple fix to the problem. Further it's used to define the correct velocity limit and establish optimal layout for the piping network.

1.0 INTRODUCTION

There are several problem areas that may not be solved by Computational Fluid dynamic calculations (CFD) or Structural Finite Element Analysis (FEA) alone. For example a study of hydrodynamic sloshing loads due to seismic activity on the walls and internals of a pressure vessel requires the input of fluid induced force or pressure from CFD to be fed as an input to the FEA calculation in order to accurately determine the locations of high stress due to the fluid force. The integration of these two distinct computer simulation technologies called Fluid-solid-interaction (FSI) is considered to be a state-of-the-art approach using conventional technologies and seen to give more accurate results compared to the traditional approach of using each methodology separately.

The use of CFD and FEA for analyzing transient thermal stresses and dynamic loads due to pressure, impact or fluid induced forces in industrial equipment and machinery is an ongoing effort. Most analysts use either CFD or FEA depending on their areas of expertise and certain assumed boundary conditions are given for either the fluid or solid part. The combination of the two fields is useful in analyzing steady state and transient phenomena such as wind blown structures, flow induced vibration where the two fields are coupled. This coupling may be defined as a one-way coupling or two-way coupling.

In a one-way coupling the temperatures or fluid loads from a CFD calculation are passed to the FEA grid for a single pass. In a two-way coupling while the forces and temperatures are passed from CFD to FEA there are also displacements and stresses passed back to the CFD and there is a continuous interaction between the two.

In Figure 1, the vortex forces are transferred to the FEA calculation and the displacement of the plate is transferred back to CFD showing the plate moving according to the vortex shedding pattern.

Another area where FSI is used is in refineries and LNG plants containing complex piping networks and pressure vessels. These large diameter pipes also contain pressure and flow control valves and originate from compressors. Flow induced vibration and noise due to these effects may be studied using FSI techniques.

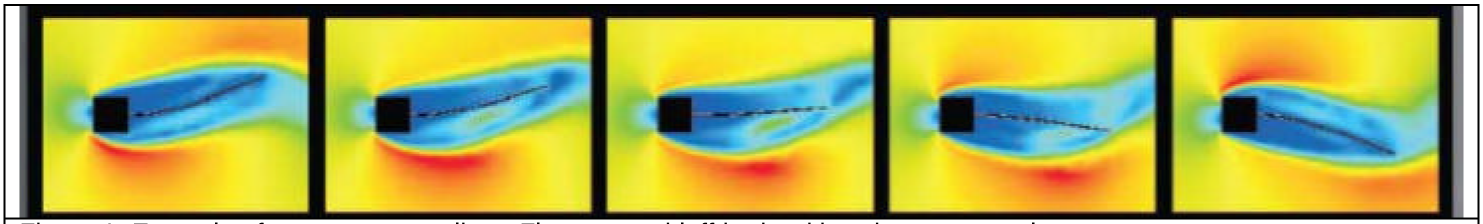

Figure 1: Example of a two way coupling. Flow over a bluff body with a downstream plate.

2.0 BASIC EQUATIONS

Computational fluid dynamics (CFD) is the method of solving the governing mass, momentum and energy equations on a computational grid representing the domain in question. CFD allows for the construction of realistic full-scale computer models, which simulate single and multiphase fluid flow and all modes of heat transfer. The CFD model accounts for the actual physical geometry of the equipment and applies fundamental physical principles to compute the temperatures, velocities, and pressures. In this paper both single and multiple phase equations are used.

2.1 Governing Equations for single and multiphase flows

Steady-state turbulent fluid flow equations are governed by the Favre averaged (density averaged) mass, momentum and scalar transport equations [Ref 2].

$$\nabla \bullet (\overline{r}\widetilde{U}) = 0 \qquad (1)$$

$$\nabla \bullet \left[\overline{r}\widetilde{U}\widetilde{f} - \Gamma_f \nabla \widetilde{f}\right] = \widetilde{S}_f \qquad (2)$$

$$\nabla \bullet \left[\overline{r}\widetilde{U} \otimes \widetilde{U} - m_{eff} (\nabla \widetilde{U} + \nabla^T \widetilde{U})\right] = -\nabla \bullet (\overline{p}\widetilde{\widetilde{G}}) + S_{\widetilde{U}} \qquad (3)$$

$\otimes$ is the tensor product operator

$\widetilde{\widetilde{G}}$ is the metric tensor

Γ_f is the diffusion coefficient

S_i is the source term

Two phase flows are solved using the Volume of Fluid (VOF) approach. This approach relies on the fact that two or more fluids or phases are not interpenetrating. The tracking of the interfaces between phases is accomplished by the solution of a continuity equation for the volume fraction of one or more of the phases [Ref. 3]. For the qth phase the equation has the form

$$\frac{\partial \alpha_q}{\partial t} + \vec{v} \cdot \nabla \alpha_q = \frac{S_{\alpha_q}}{\rho_q} \qquad (4)$$

where,

α_q = The q'th fluid's volume fraction.

$\vec{v}$ = overall velocity vector
ρ = density
$S_{\alpha q}$ = Source Term

A single momentum equation is solved through the domain and the resulting velocity field is shared among phases.

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot \left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}\right)\right] + \rho\vec{g} + \vec{F} \tag{5}$$

Where

$-\nabla p$ - = Pressure term
$\nabla \cdot \left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}\right)\right]$ = Viscous term
$\rho\,\vec{g}$ = Gravitational acceleration term
$\vec{F}$ = Force vector

2.2 Governing equations for stress analysis

Structural analysis is the most common application of the finite element Analysis [Ref 4]

The stress is related to the strains by

$$\{\sigma\} = [D]\{\varepsilon^{el}\} \tag{6}$$

Where

{s} is the stress vector = $\lfloor \sigma_x\ \sigma_y\ \sigma_z\ \sigma_{xy}\ \sigma_{yz}\ \sigma_{xz} \rfloor^{T}$
[D] = stress-strain matrix
$\{e^{el}\}$ is the elastic strain vector made with the components $\lfloor \varepsilon_x\ \varepsilon_y\ \varepsilon_z\ \varepsilon_{xy}\ \varepsilon_{yz}\ \varepsilon_{xz} \rfloor^{T}$

The von Mises or equivalent stress s_e is computed as follows

$$s_e = (\frac{1}{2}\left[(s_1 - s_2)^2 + (s_2 - s_3)^2 + (s_3 - s_1)^2\right])^{\frac{1}{2}} \tag{7}$$

3.0 Study of Hydrodynamic Sloshing Loads

This study compares the stresses generated in a pressure vessel and internals due to fluid sloshing forces under seismic activity. The traditional ASCE 4-98 [Ref 5] code-based approach of using spring elements and lumped mass elements to simulate sloshing is compared with fluid-solid interaction methodology.

In the ASCE 4-98 approach, the fluid contents of the vessel are divided into two classes: impulsive mass and convective mass. The impulsive mass is the lower portion of the liquid contents that is known to move with the vessel wall below the sloshing level. Figure 2 shows a representation of the spring-mass model while Figure 3 shows the model of the pressure vessel along with internal components and support structures.

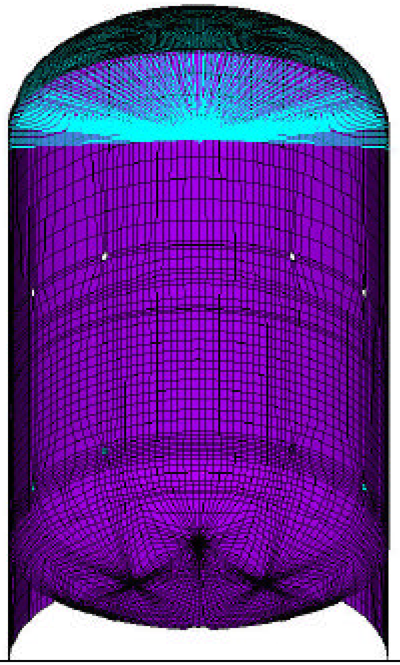

Figure 2 A pressure Vessel with spring elements and lumped mass used by ASCE 4-98 approach

The convective mass is the sloshing component of the liquid, which is comprised of a central node representing the sloshing convective mass fraction to account for the global effect of sloshing on the vessel. Local effects due to the sloshing fluid are not accounted for using this model.

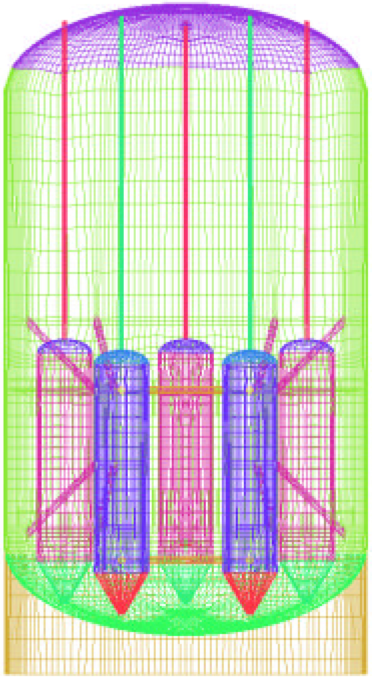

Figure 3: Pressure Vessel with internal components and supports used for the comparison study.

Dynamic analysis of earthquakes requires the in-structure response spectra (ISRS) for response spectrum analysis in each orthogonal direction denoted by E-W, N-S and the vertical directions. These are normally given in the frequency domain and contain curves for various damping ratios at different frequencies. The data may be used for ground acceleration effects using modal and spectral analysis. For CFD and stress calculations in the time domain, the ISRS curves may be converted to a time-history using Fourier transformations. For this comparative study a damping ratio of 0.5% was used in all directions for consistency.

Figure 4 shows the sequence of steps followed by the FSI approach to obtain the maximum stress due to both ground accelerations and the sloshing of the fluid. CFD was used to study the sloshing of the free surface under seismic loads given by the time history acceleration curves. The sloshing of the liquid slurry exerts forces on the wall of the vessel and the internals. These forces are given by the pressure of the fluid acting on the walls and may be derived from fluid dynamic calculation results. Multiphase flow capability was used to study both the impulsive and convective mass of liquid moving on the free surface. Gravity loads contributed to the hydrostatic head based on the liquid heights and hydrodynamic forces acting on the walls and internals are also taken into account by the calculations.

The resulting sloshing forces on the vessel and the internals from CFD were then superimposed on a mesh for a transient Finite Element analysis in the time domain to determine local response. On the other hand the vessels effects due to ground and foundation movements were analyzed in the frequency domain using Modal and spectral analysis. The dominant frequencies are combined using the square root of the sum of squares (SRSS) method to obtain the maximum stress intensity. Finally, the localize stresses due to sloshing were combined with the vessel's global response from ground acceleration to obtain the total stress intensity.

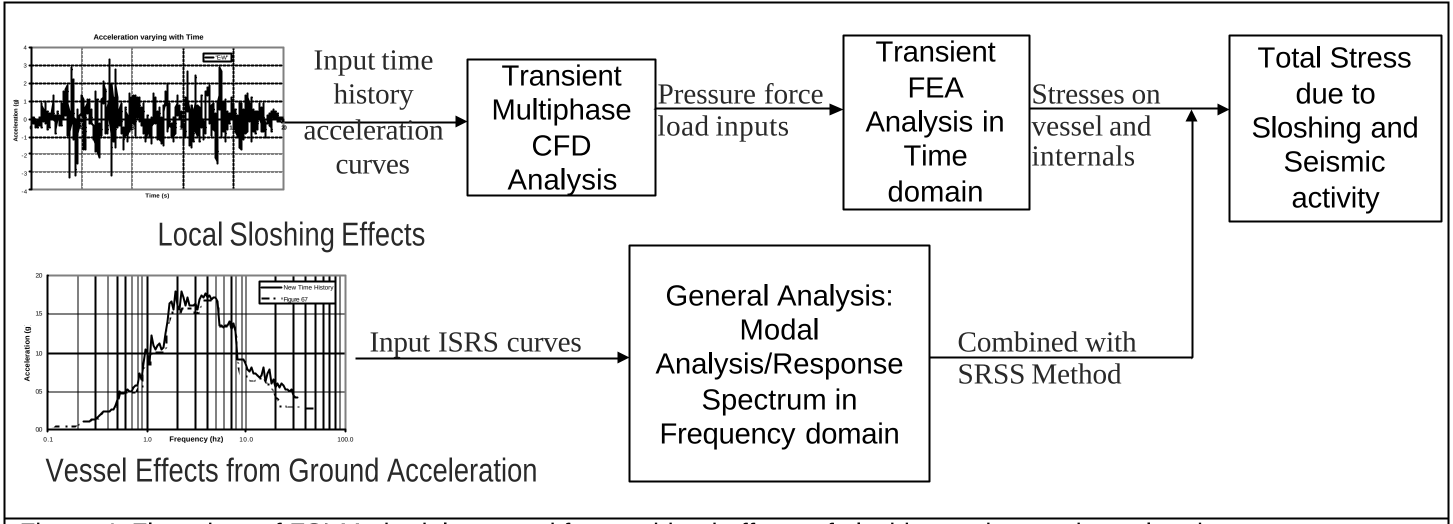

Figure 4: Flow chart of FSI Methodology used for combined effects of sloshing and ground accelerations

Four cases were studied

1. Full Liquid State: Vessel is full of liquid up to the tangent line
2. Half-Full Liquid State: Vessel is half full of liquid
3. Near Empty Liquid State: Vessel is nearly empty with liquid height to the mid-height of all internal components.
4. Using the Vessel Full liquid state various diameter vessels.

The allowable load criteria is from ASME for S=20KSI

SS 3XX Allowable	Variable Name	Operating	Seismic
Maximum Stress (primary membrane)	P_m	P_m <= 20ksi	P_m <= 24ksi
Combined Stress (local membrane or membrane + bending)	P_L, P_b	P_L <= 30ksi and P_b <= 30ksi	P_L <= 36ksi and P_b <= 36ksi
Secondary Stress (membrane + bending)	Q	Q <= 60ksi	Q <= 60ksi

3.1 CFD results

Certain results may be formulated from the CFD calculations alone. When the vessel is full of fluid, the net force acting on the vessel walls is more than on the internals. The long inlet pipes also experience a significant lateral force applied in the direction of the sloshing fluid. The fluid sloshing motion for full vessel at 3 seconds of seismic activity is shown in Figure 5. The free surface is colored by magnitude of velocity.

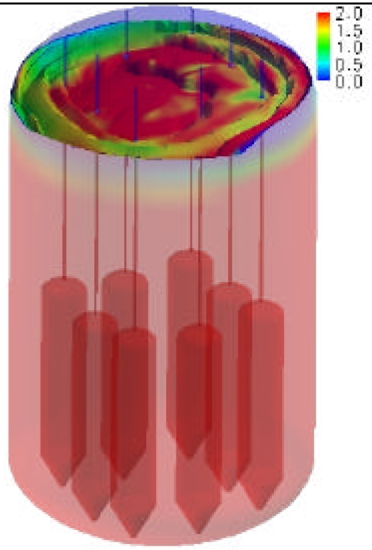

Figure 5: Fluid sloshing in "vessel full" case after 3 seconds of seismic activity

The variation of the pressure over time is shown in Figure 6. The pressures are calculated as the area-weighted averages taken over portions of the fluid which are exposed to the free surface.

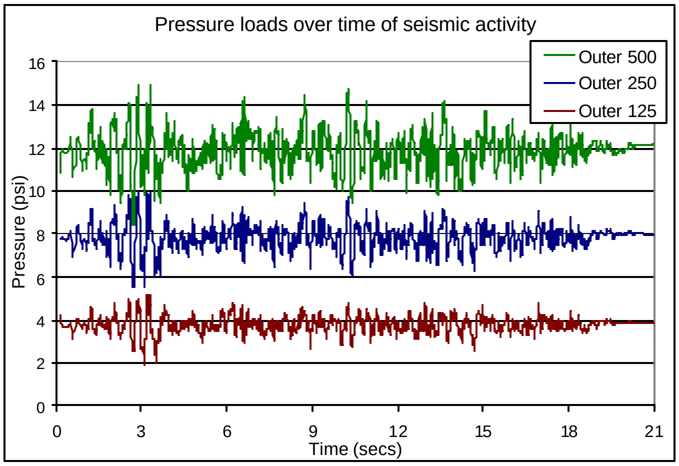

Figure 6: Pressure loads for various diameter vessels

It may be noted that for this series of ISRS curves the maximum pressures are observed around 3 seconds of seismic activity.

A Fourier analysis of the pressure curves reveals the dominant frequencies for each of the E-W, N-S and vertical directions. A dominant frequency of 2.5Hz is seen in the vertical direction from Figure 7.

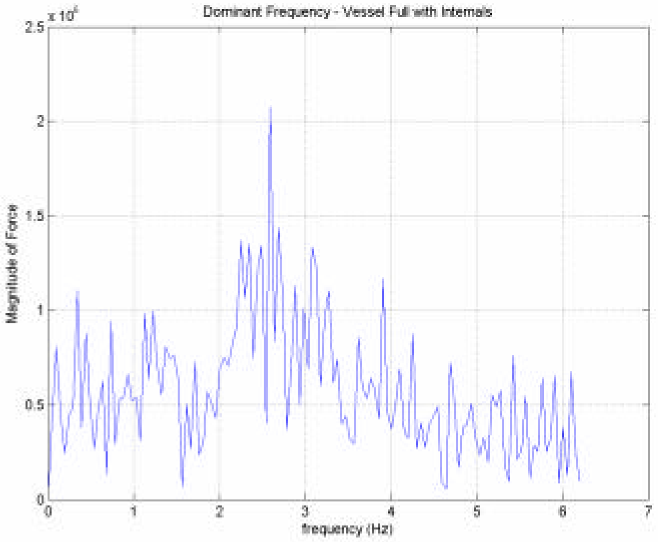

Figure 7: Dominant frequencies for the vertical force

3.2 FEA Calculations:

The pressure data from the CFD calculations were analyzed and it was observed that the maximum load and load variation appears between 2.8 and 3.0 seconds of seismic activity. Therefore the FEA transient calculations were performed from 2 seconds to 4 seconds. The loads on the walls and internals of the CFD calculations were applied to the nodes of a similar mesh in the FEA calculations.

The maximum stress intensity using the FSI approach are shown for vessel exterior (figure 8) and on internal components (figure 9) for a vessel full case.

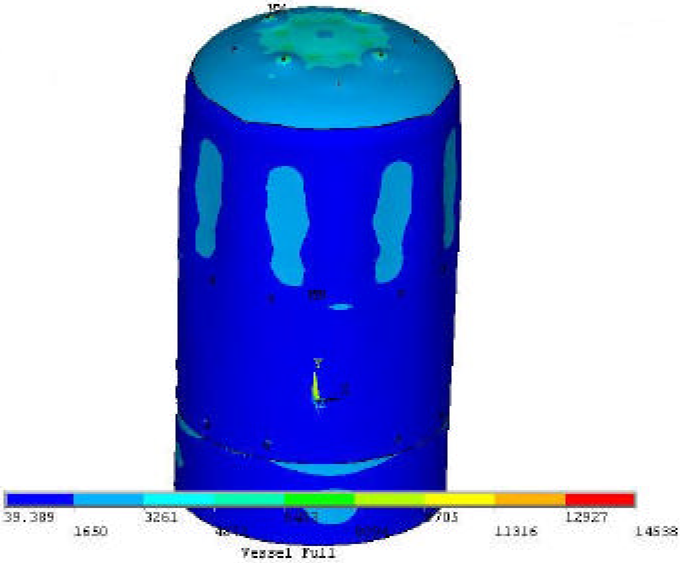

Figure 8: Max stress intensity outer surface

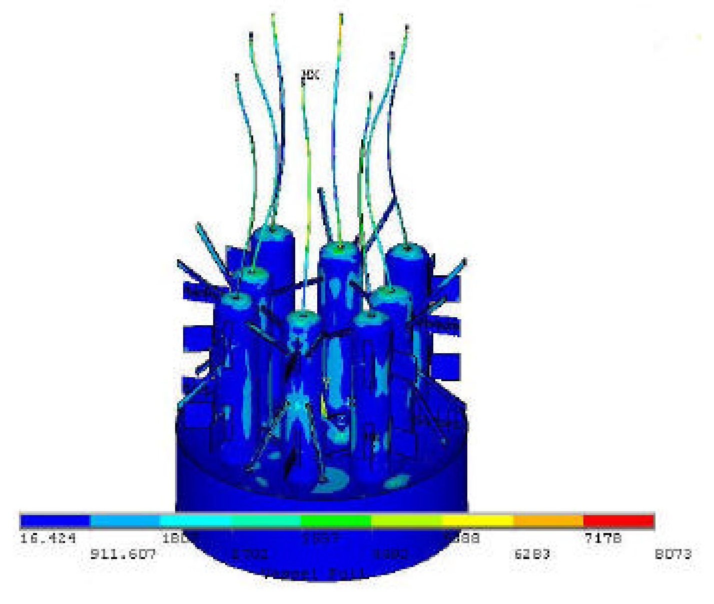

Figure 9: Stress intensity contours on internal components

3.3 Results of the analysis

Some typical values of stress intensities obtained using the Code-based- approach and comparison with the FSI results are shown in Figure 10. The results show that Sloshing loads due to seismic activity is not significant compared to other loads the vessel may experience. The total increase in stress intensity is less than 10% of the allowable stress. The local sloshing effects combined with the global vessel response to seismic ground motion show the increase in maximum stress intensity to be 2-3%. The total pressures due to sloshing are greatest for a full vessel and the forces go down as the liquid level decreases. Subsequently the forces are highest at the free surface.

When comparing the effects of sloshing on vessel internals, the more accurate FSI approach is within 5% of the stresses determined using the traditional code-based approach. Therefore the code-based approach may be followed for most seismic studies and typical vessel design because using FSI for every vessel is very time intensive and computationally expensive.

		Code-Based		**FSI Approach**				
		Ground Acceleration		**Sloshing Only**			**GA + Sloshing**	
Model	Lumped Mass Sloshing Loads	x						
	General Analysis		x	x	x	x	x	x
Liquid State	Full Vessel	x	x	x			x	
	Half Full Vessel				x			x
	Nearly Empty Vessel					x		
Loads	Ground Accelerations Only	x	x				x	x
	CFD Sloshing Loads			x	x	x	x	x
Analysis Method	Modal/Response Spectrum	x	x				x	x
	Transient Time History			x	x	x	x	x
Maximum Stress Intensity (ksi)	**Top Surface**	38.1	38.1	8.1	6.8	6.1	38.9	38.7

Figure 10: Typical values of stress intensities obtained with this analysis based on the given ISRS curves

4.0 Vibration in pipes and line sizing criteria

Piping downstream of compressors is subject to vibration. The unsteady swirling flow creates torsion in the pipeline. The pulsating flow causes flow induced noise, vibration and lead to fatigue failures.

Depending on the number of compressors the unsteady flow rates cause several problems. After they lead to the main header, they may be a bottle neck to excess flow rate if all the compressors are working in phase for example. The main stressed areas are the tee-joints, pipe bends, and welds at the shoes, joints and support locations.

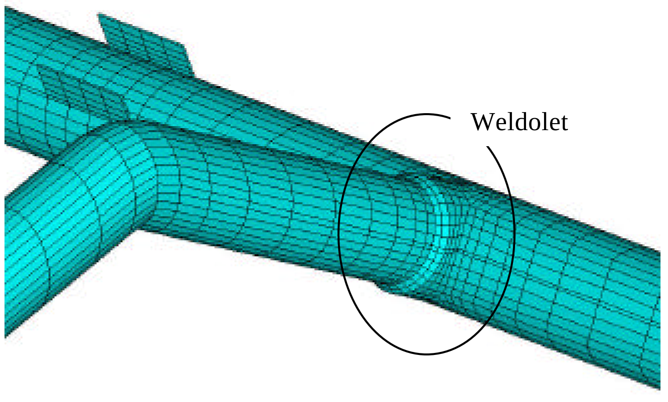

Figure 11: Initial configuration–10" laterals to 12" header

The diameters of the laterals from the compressors and the main header need to chosen with care to avoid overstress.

FSI may be used to study these problems and establish fast design changes that enable the selection of the best piping configuration in a short time. This is mainly due to the fact that parametric techniques may be used to change diameters, layout, or use alternate lines and help optimize the flow field.

This problem deals with flow from two compressors with 10” diameter laterals leading to a 12” diameter main header (Figure 11). The laterals are at 45 degree angles to the header with the tee-joint a weldolet. The expansion of a plant needed an additional compressor to operate in addition to the existing ones. Due to the increased flow rates the existing lines were overworked. In the figure the swirling flow from the compressors may be observed (Figure 12). Further, down stream of the last compressor the flow rate is seen to create a necked flow.

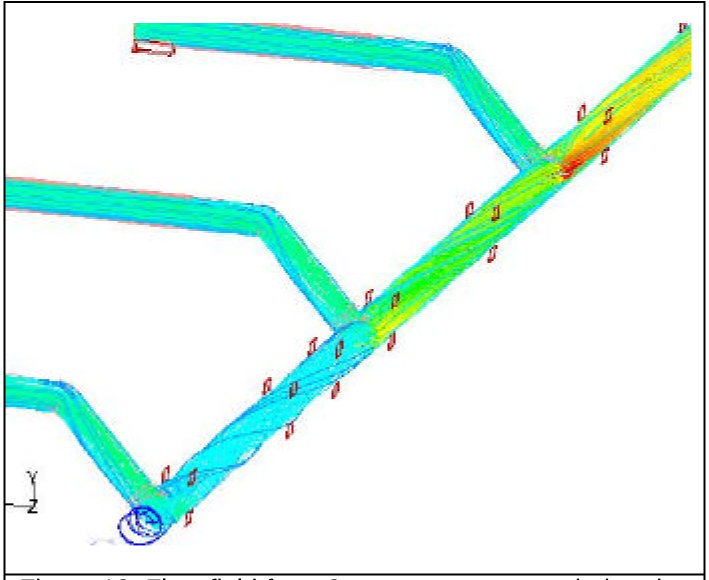

Figure 12: Flow field from 3 compressors to main header

While erosion velocity limits are about 100m/s, it was uncertain whether the tees, lines, joints, supports and bends would be overstressed with the increased velocities.

4.1 FSI Analysis

Following are the objectives of the Analysis:

- Observe the velocity and pressure field using the existing configuration.
- Calculate the stresses in components with the existing configuration
- Based on stress results make design recommendations and rerun analysis with the design changes implemented.
- Determine if the velocity limit guidelines hold in the current design and to define the correct velocity limit and establish an optimal layout for the piping system.

The FSI methodology followed was as follows.

- Run CFD analysis and plot the flow field and streamlines.
- The fluid analysis may be single phase or multiphase depending on the amount of liquid flowing in the pipelines
- Further to obtain an alternating stress for fatigue and vibration, an unsteady calculation may be performed
- Obtain fluid pressures on the piping system
- Superimpose the pressure field as loads into the FEA analysis
- Run static or dynamic analysis to obtain stresses in the network of pipes.

From the flow field shown in Figure 12, the pressures are given as loads to the Finite Element Analysis.

It is observed that with the existing configuration, the lines are overstressed at a velocity over 100m/s.

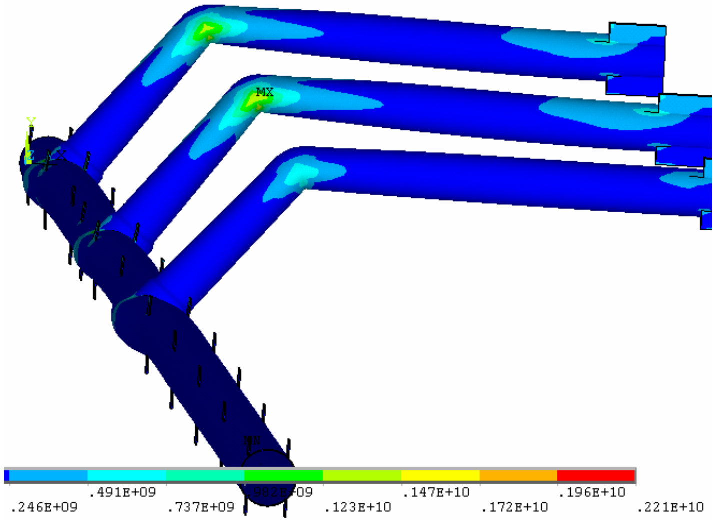

Figure 13: Stress contours due to flow from compressors

The stress intensity contours (Figure 13) show the main areas of overstress

- cold shoes welded to the pipes (adjacent to the laterals),
- weldolet and butt weld at the Tee and
- the 45 degree bend joint in the laterals

The cheapest optimal solution using a few design changes parametrically was found using the FSI methodology. It was observed that most stresses could be mitigated if the last 15 feet of the laterals was increased from 5” to 6” diameter (Figure 14). This allows for smoother flow transition and reduces the stresses in the joints.

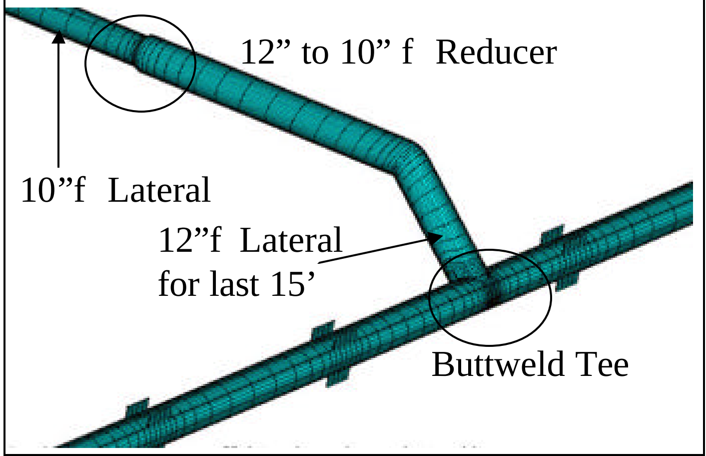

Figure 14: Final configuration mitigating overstress

Therefore FSI studies enable safe operation of piping Networks and helps establish optimal velocity and flow rate guidelines. FSI proved to be an effective tool to study the velocity limits and provide a cheap alternative solution to the problem.

5.0 CONCLUSIONS

Two applications are chosen to exemplify the importance of using FSI as a cost effective tool for studying fluid flow related stresses in pipes and vessels.

These analysis tools can also be used for finding root causes of some industrial problems and provide fixes for the same through a correct engineering approach. The same procedures and tools may be used as a design guide for projects thus showing strong business development potential. Following the guidelines reduces schedule and accelerates project closure.

Using these analysis methodologies has obviated the use of costly equipment for experimentation and instruments and the operators needed for this task. CFD and FEA strive to be tools that can immensely help most industries.

The FSI approach allows for innovative application of state-of-the-art computational methods to improve realism and accuracy in our engineering analyses. Integration of these two analysis methods (CFD/FEA) is a valuable tool for risk reduction and enhanced accuracy, which could lead to cost savings by potentially reducing design conservatism, as well as have predictive value for safety and schedule. In future, large projects will likely involve more first-of-a-kind designs and unconventional structures and materials. Performing detailed fluid-structural analysis of these projects would be a task of critical importance.

In conclusion the benefits of using FSI

- Useful recommendation made for future projects to avoid problems and ensure safe operation
- Fast turnaround without resorting to expensive experimentation
- Fast design changes using parametric techniques enable optimizing and selection of best configuration in the shortest time
- Increased accuracy, lesser conservatism in solution of unique problems

6.0 REFERENCES

1. Diwakar P.M & Richardson F, “Mitigation of bending stress and failure due to temperature differentials in piping systems carrying multiphase fluids – using CFD and FEA”, IMECE 2005, November 2005.

2. Crane Company Engineering Division, “Flow of Fluids through Valves, Fittings, and Pipe” Technical Paper No. 410. Chicago, IL. 1969.

3. S.A.Vasquez and V.A.Ivanov, “A phase coupled method for solving multiphase problems on unstructured meshes”, Proceedings of ASME FEDSM’00: ASME June 2000.

4. Irons, B. M., "A Frontal Solution Program for Finite Element Analysis", International Journal for Numerical Methods in Engineering, Vol. 2, No. 1, January, 1970, pp. 5-23 (Discussion May, 1970, p. 149).

5. Seismic Analysis of Safety-related nuclear structures and commentary”, 1998, ASCE 4-98.