<Technical Notes>

IMPACT ANALYSIS OF A WATER STORAGE TANK

MYUNG JO JHUNG*, JONG CHULL JO and SANG JIN JEONG[1]
Korea Institute of Nuclear Safety
19 Guseong-dong, Yuseong-gu, Daejeon, 305-338 Korea
[1]TaeSung S&E Inc.
7-28 Sinchun-dong, Songpa-gu, Seoul, 138-735 Korea
*Corresponding author. E-mail : mjj@kins.re.kr

Received October 14, 2005
Accepted for Publication July 18, 2006

This study investigates the dynamic response characteristics of a structure impacted by a high speed projectile. The impact of a 300 kg projectile on a water storage tank is simulated by the general purpose computer codes ANSYS and LS-DYNA. Several methods to simulate the impact are considered and their results are compared. Based upon this, an alternative impact analysis method that is equivalent to an explicit dynamic analysis is proposed. The effect of fluid on the responses of the tank is also addressed.

KEYWORDS : Projectile, Impact Analysis, Water Storage Tank, Fluid-Structure Interaction

1. INTRODUCTION

High speed projectile impacts upon structures can cause structural damage resulting in catastrophic failure, including huge loss of life. Typical examples involve aircraft, which are particularly vulnerable to high speed projectile impact, resulting from uncontained engine debris following engine failure, objects encountered during flight such as bird strikes or hail and impact with runway debris during take off or landing [1].

In addition, similar risks are posed to safety critical structures in many other industries such as rail, chemical, nuclear, and offshore industries. One example occurred on April 16, 2005 at Kori nuclear power plant unit 1 during the 23rd outage when the piping system of a 300 kg connecting main steam line to the silencer impacted the refueling water storage tank [2]. The tank was damaged with maximum permanent deformation of 60 mm in the radial direction.

It is therefore becoming increasingly important to be able to model and predict such phenomena. If the structure of interest is in contact with fluid, complex fluid structure interactions produce structural behavior that is often difficult to predict. Understanding of this behavior is needed to assess both the suitability and predictive capabilities of numerical modeling techniques such as finite element analyses for the analysis of transient fluid structure interactions.

In the present study, a numerical analysis of a water storage tank impacted by a high speed projectile is performed. Two different general purpose computer programs, ANSYS [3] and LS-DYNA [4], are used and their response characteristics are investigated for the impact analysis. The effect of fluid on the impact responses is also addressed. An alternative analytical method to predict the responses due to the impact using ANSYS is proposed and verified to be useful.

2. ANALYSIS

2.1 Modal Analysis

The water storage tank consists of a shell and cover, as shown in Fig. 1, which are made of stainless steel 321. The physical properties of the material at ambient temperature are as follows: Young's modulus = 195E9 Pa, Poisson's ratio = 0.3, and mass density = 8027 kg/m^3. Water is used as the contained fluid, having a density of 996 kg/m^3. The sound speed in water is 1486 m/sec, which is equivalent to a bulk modulus of elasticity of 2.2 GPa.

Finite element analyses using the commercial computer code ANSYS 9.0 [3] are performed to obtain the modal characteristics of the tank. A three-dimensional model having 54418 nodes and 51436 elements is developed for the water storage tank with fluid, as shown in Fig. 2. The fluid region is divided into a number of 3-dimensional contained fluid elements (FLUID80) with eight nodes having three degrees of freedom at each node. The fluid element FLUID80 is particularly well suited for calculating hydrostatic pressures and fluid/solid interactions. The tank is modeled as 4-node plastic large strain shell elements (SHELL43).

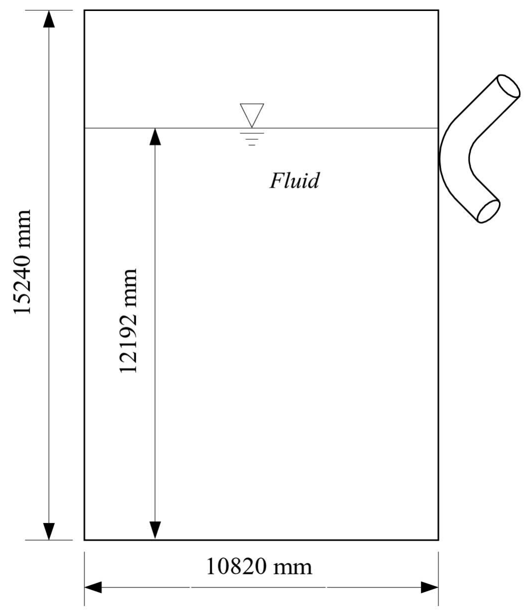

Fig. 1. Impact of Missile Pipe on the Tank

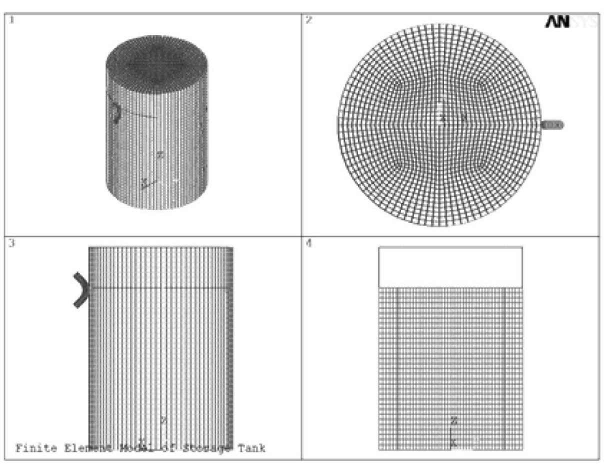

Fig. 2. Finite Element Model of Water Storage Tank

The bottom nodes of the shell are fixed in all six degrees of freedom. The fluid movement at the bottom of the tank is considered to be constrained in the vertical direction. The vertical velocities of the fluid element nodes adjacent to each surface of the wetted shell coincide with those of shell.

The Block Lanczos method is used for the eigenvalue and eigenvector extractions to calculate 300 frequencies including the fluid modes [4]. This method uses the Lanczos algorithm, where the Lanczos recursion is performed with a block of vectors. This method is as accurate as the subspace method, but faster. The Block Lanczos method is especially powerful when searching for eigenfrequencies in a given part of the eigenvalue spectrum of a given system. The convergence rate of the eigenfrequencies will be roughly the same when extracting modes in the midrange and higher end of the spectrum as when extracting the lowest modes.

2.2 Impact Analysis

The piping system impacts the water storage tank as shown in Fig. 1. Two types of impact for the ANSYS code are used to obtain the response characteristics of the tank: one uses the initial velocity IC with the contact and target elements (Fig. 2) and the other applies an impulse that is equivalent to the impact. The impact force applied is calculated as follows:

$$F_{impulse} = \frac{m \times v}{t} \tag{1}$$

where m is the mass of the projectile, v the impact velocity, and t the impact duration. The impact duration is usually considered to be about 0.001 second depending on the impact type. The impact force is applied to nodes at a certain area, which is characterized by the shape of the projectile. For the piping system the force is assumed to be normal-distributed and the center of the area maintains the maximum value of the force.

The general purpose explicit dynamic finite element program LS-DYNA is also used to obtain the response characteristics of the tank. The acceleration evaluated at time t is

$$\{a_t\} = [M]^{-1}\left([F_t^{ext}] - [F_t^{int}]\right) \tag{2}$$

where $[F_t^{ext}]$ is the applied external and body force vector and $[F_t^{int}]$ is the internal force vector, which is given by

$$F^{int} = \sum\left(\int_{\Omega} B^T \sigma_n d\Omega + F^{hg}\right) + F^{contact} \tag{3}$$

where F^{hg} is the hourglass resistance force and $F^{contact}$ is the contact force. The velocities and displacements are then evaluated:

$$\{v_{t+\Delta t/2}\} = \{v_{t-\Delta t/2}\} + \{a_t\}\Delta t_t \tag{4}$$

$$\{u_{t+\Delta t/2}\} = \{u_t\} + \{u_{t+\Delta t/2}\}\Delta t_{t+\Delta t/2} \tag{5}$$

where $\Delta t_{t+\Delta t/2}$ is 0.5($\Delta t_t + \Delta t_{t+\Delta t}$) and $\Delta t_{t-\Delta t/2}$ is 0.5($\Delta t_t - \Delta t_{t+\Delta t}$). The geometry is updated by adding the displacement increments to the initial geometry $\{x_0\}$;

$$\{x_{t+\Delta t/2}\} = \{x_0\} + \{u_{t+\Delta t}\} \tag{6}$$

ANSYS LS-DYNA interface seamlessly links ANSYS pre- and post-processing software with the LS-DYNA explicit solver. Therefore, the ANSYS input deck is used with the exception of using the explicit thin structural shell SHELL163 and the explicit 3-dimensional structural solid SOLID164 instead of SHELL43 and FLUID80 elements for the tank and fluid, respectively.

3. RESULTS AND DISCUSSION

Several different cases were analyzed in order to obtain the modal and impact characteristics of different models: without- and with-fluid, elastic and elastic-plastic, ANSYS and LS-DYNA, impact types of initial velocity and impact force, and fluid-structure interaction representation of LS-DYNA.

The frequency comparisons between without- and with-fluid are shown in Fig. 3. Because the tank is only 80% filled, the cover does not contact the fluid. Therefore, the frequencies of the cover plate are not changed due to the fluid. The frequencies of the shell are summarized in Fig. 3 for the first 2 axial modes. The effect of the fluid on the frequencies of a circular shell wetted with fluid can be assessed using the normalized frequency, which is defined as the natural frequency of a structure in contact with a fluid divided by the corresponding natural frequency in a vacuum. The normalized natural frequencies have values between one and zero due to the added mass effect of the fluid. Fig. 4 shows the normalized natural frequencies for the shell modes containing fluid. As the number of circumferential modes increases, the normalized natural frequencies increase by the gradual reduction of the relative added mass effect [6-8]. Therefore, an increase of the circumferential mode causes an increase in the normalized natural frequencies for all cases of modes except (1, 1) mode, which is a typical case of a shell with a cover plate. Typical mode shapes of the water storage tank without water are shown in Fig. 5 and those of the cover are shown in Fig. 6.

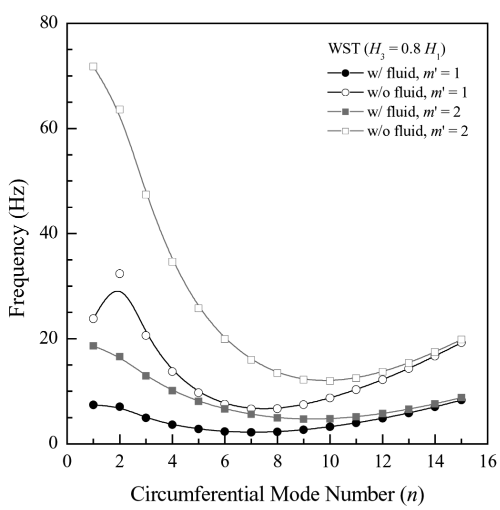

Fig. 3. Frequency Comparisons Between with- and Without-Fluid Cases

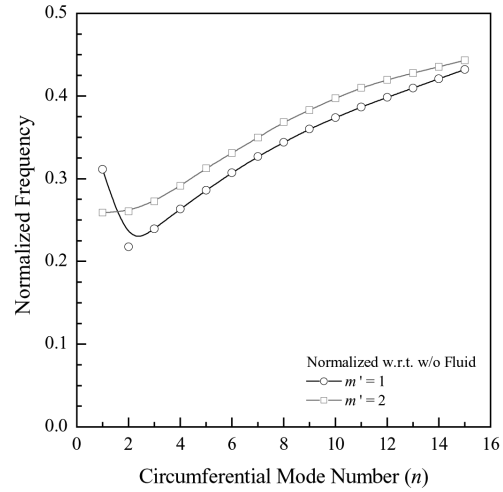

Fig. 4. Normalized Frequencies of Tank with Fluid with Respect to Without-Fluid

An impact force that will be equivalent to a projectile with an impact velocity can be calculated using Eq. (1) and it is applied to the nodes of the impact location. In this case the impact duration must be known and the effect of the impact duration is investigated. If a 300 kg projectile hits the tank with a velocity of 10 m/sec, the impact force applied will be 3,000,000 N for an impact duration of 0.001 second. This will be 1,500,000 N and 1,000,000 N for 0.002 and 0.003 second, respectively. Displacement time histories of the impact point according to the impact duration are shown in Fig. 7. The maximum values of deflection do not vary with the values of the impact forces. By comparing the displacement time histories between the transient analysis results using the impact velocity and impact force in Fig. 8, it is found that an impact duration of 0.003 second provides the best simulation of the impact.

There is a good agreement between ANSYS and LS-DYNA for the displacement time histories of the impact point, as shown in Fig. 9. This demonstrates that ANSYS can be used for the impact analysis instead of the general purpose explicit dynamic finite element program LS-DYNA. However, care should be taken in selecting the integration time step so that the system will not be diverged.

The system is stable in LS-DYNA if the time step size is smaller than the critical time step size;

$$\Delta t \le \Delta t^{crit} = \frac{2}{\omega_{\max}} \tag{7}$$

where $\omega_{\max}$ is the largest circular frequency. The critical time step size is automatically calculated by LS-DYNA. It

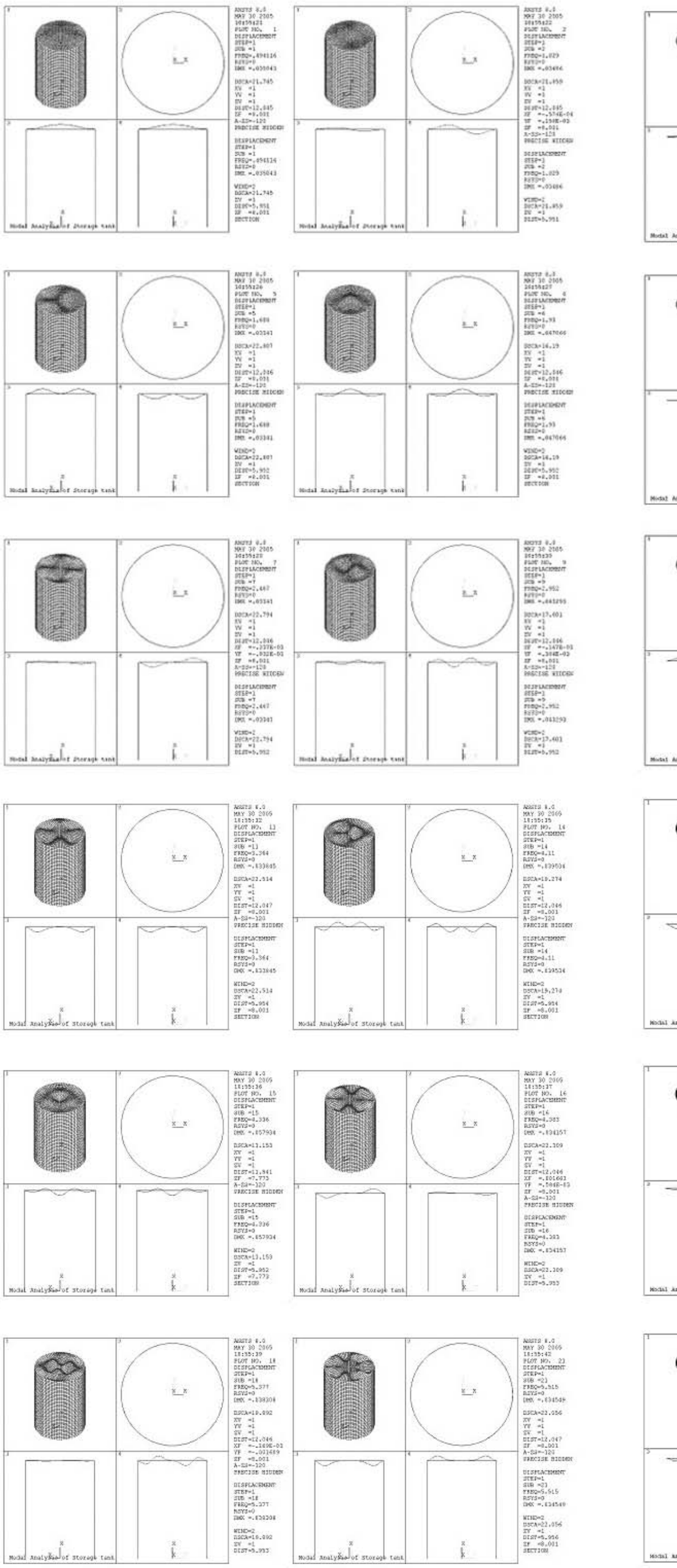

Fig. 5. Typical Mode Shapes of Water Storage Tank without Fluid

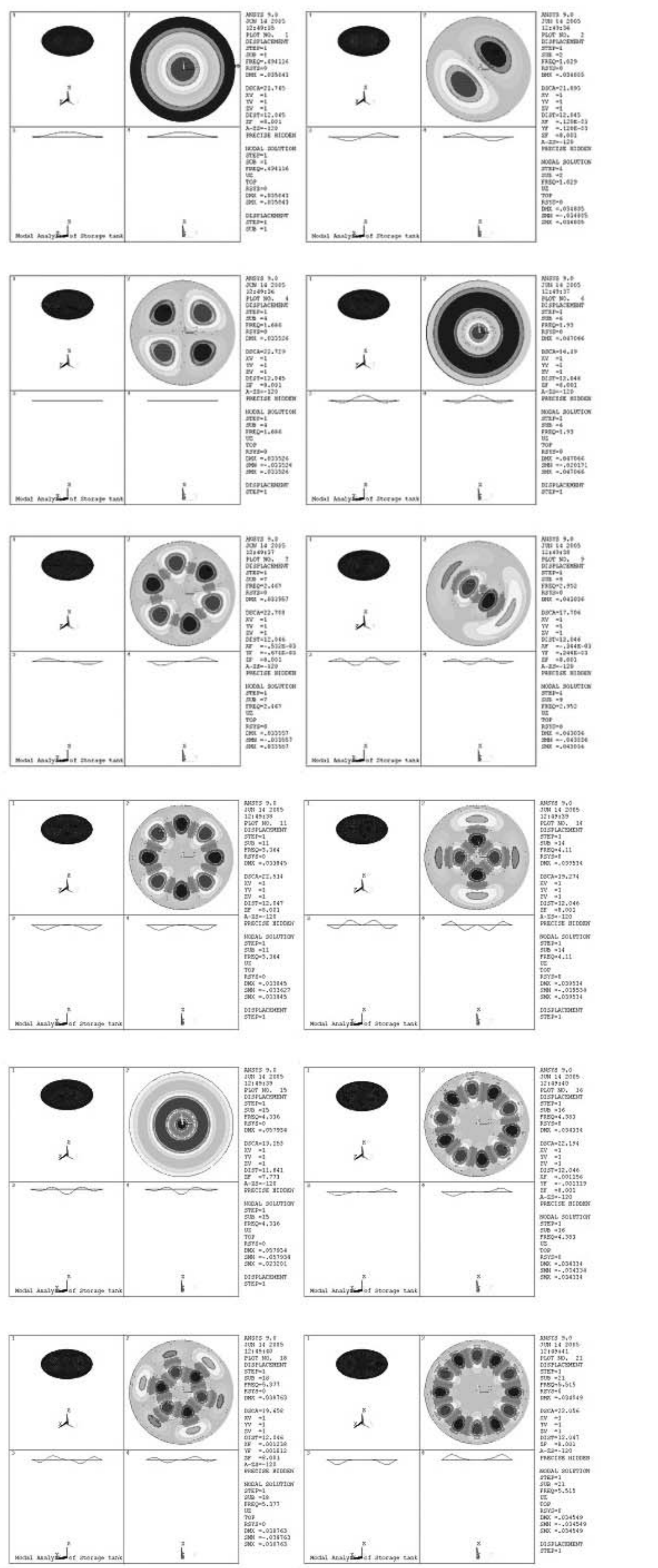

Fig. 6. Typical Mode Shapes of Cover of the Water Storage Tank

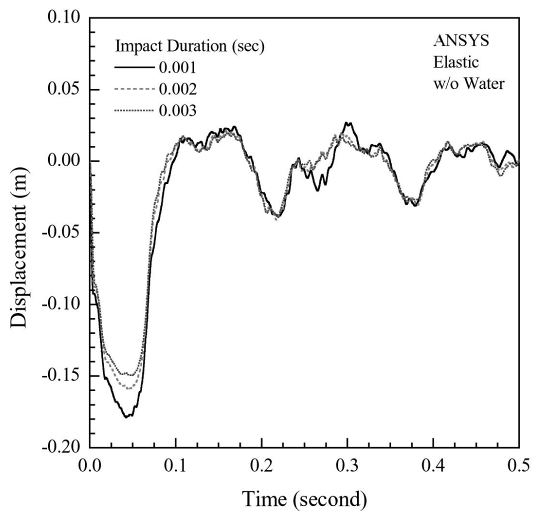

Fig. 7. Displacement Time Histories of Impact Point According to Impact Duration

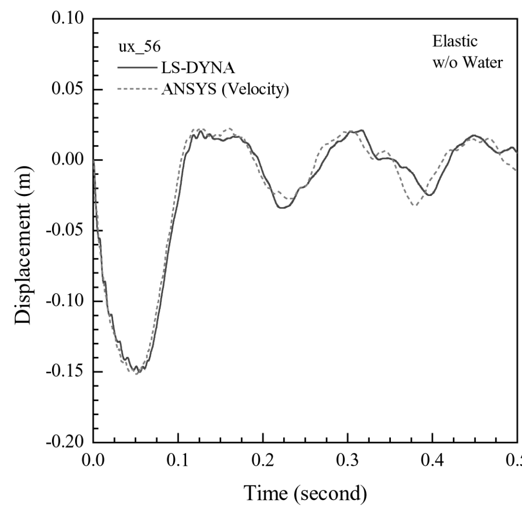

Fig. 9. Displacement Time Histories of Impact Point According to Computer Code

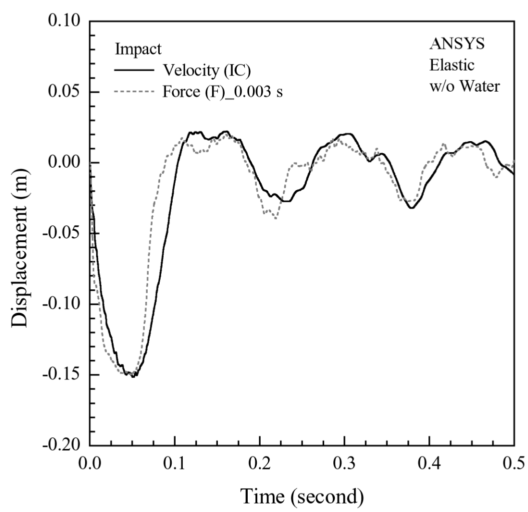

Fig. 8. Displacement Time Histories of Impact Point According to Impact Type

depends on element lengths and material properties (sonic speed). LS-DYNA checks all elements when calculating the required time step. For stability reasons a scale factor of 0.9 is used to decrease the time step:

$$\Delta t = 0.9\frac{l}{c} \tag{8}$$

where l is the characteristic length and c is the wave propagation velocity, and they are dependent on the element type. For a shell element, l is determined as the area of the element divided by the maximum length of the sides while c is

$$c = \sqrt{\frac{E}{\rho(1-\nu^2)}} \tag{9}$$

where E is Young's modulus, ρ density and ν Poisson's ratio.

For complicated problems including fluid, the fluid-structure interaction modeling sometimes requires a very fine mesh. In such a case a very small time step is needed in the ANSYS transient analysis. However, this is almost impossible due to the difficulty of convergence. Except for special cases, most problems can be solved using ANSYS instead of LS-DYNA.

The response time histories of impact point for a LS-DYNA elastic analysis without water are shown in Fig. 10 and those with water are shown in Fig. 11. The fluid-structure interaction is simulated using NROTATE and CP commands in ANSYS. The fluid and shell nodes are coupled in the radial direction for the vertical motions of the fluid element nodes adjacent to each surface of the wetted shell to coincide with those of shell nodes. However, in LS-DANA the command NROTATE does not work, and if CP command is used it is coupled in the global direction. As an alternative, coupling in two horizontal directions may be used; however, this will generate overly conservative results. Another method is to use contact elements for the fluid and shell. The responses for this method are shown in Fig. 12, and they are found to be more realistic because the fluid can move on the shell surface freely. The response time histories of the impact point between the fluid structure interaction representations are compared in Figs. 11 and 12. Also, comparisons of the responses between the without- and with-fluid cases are shown in Figs. 13 and 14. The water is expected to absorb most of the impact and the deflection decreases significantly with the inclusion of the water in the tank.

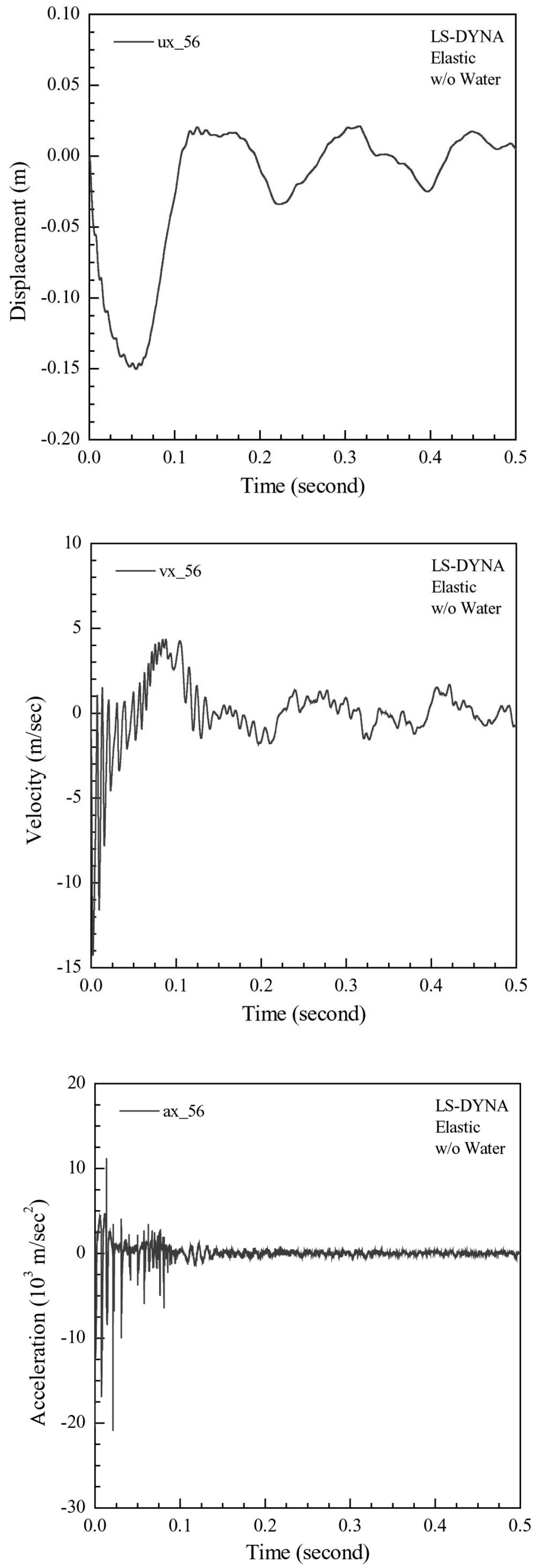

Fig. 10. Response Time Histories of Impact Point for LS-DYNA Elastic Analysis without Water

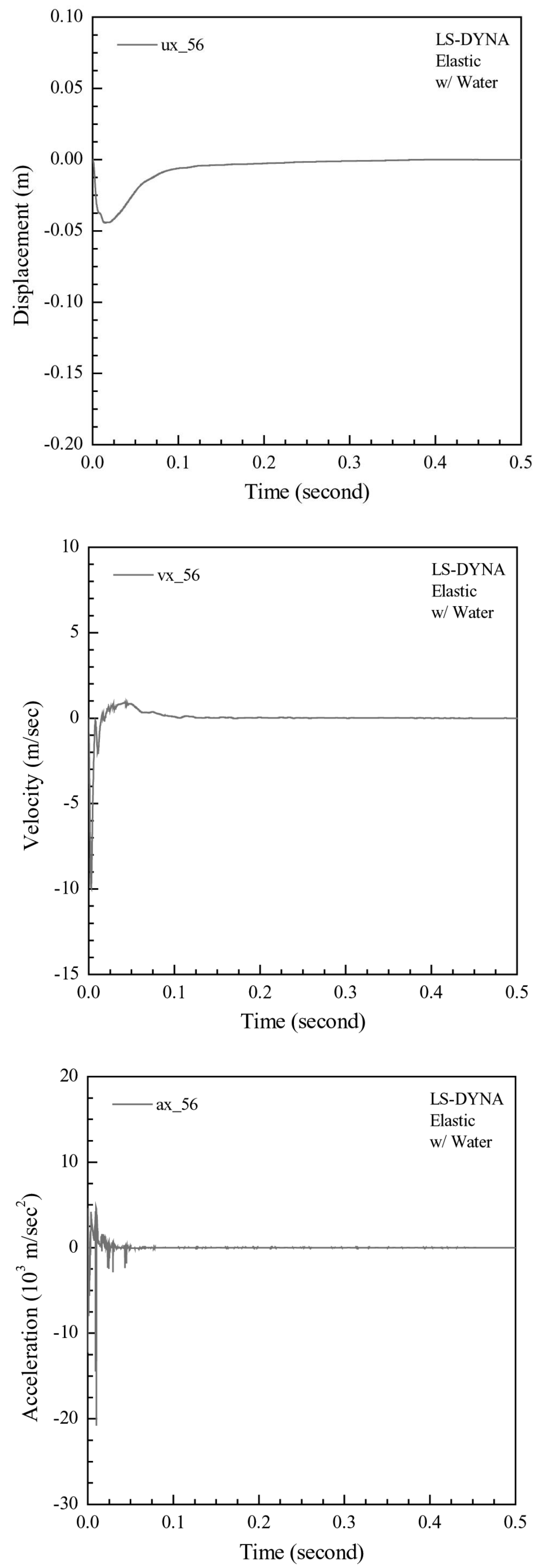

Fig. 11. Response Time Histories of Impact Point for LS-DYNA Elastic Analysis with Water (by Coupling Fluid with Shell using CPINTF)

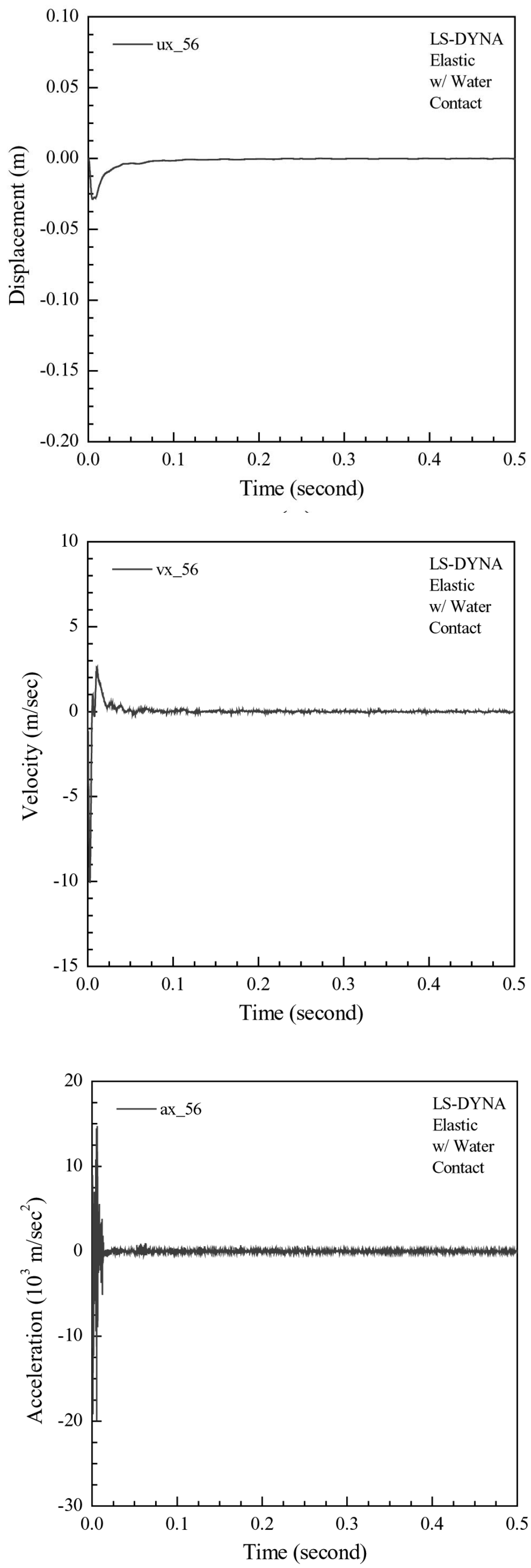

Fig. 12. Response Time Histories of Impact Point for LS-DYNA Elastic Analysis with Water (using Contact Element for Fluid and Shell)

The response comparisons between the elastic and elastic-plastic analysis are shown in Figs. 15 through 17 for displacement, acceleration, and equivalent stress time histories, respectively. As expected, the displacement and acceleration time histories are almost the same but the equivalent stresses for the elastic-plastic analysis are much lower than those of the elastic analysis.

4. CONCLUSIONS

Impact analyses of a water storage tank were performed using two general purpose computer programs, ANSYS and LS-DYNA. Several methods to simulate the impact were

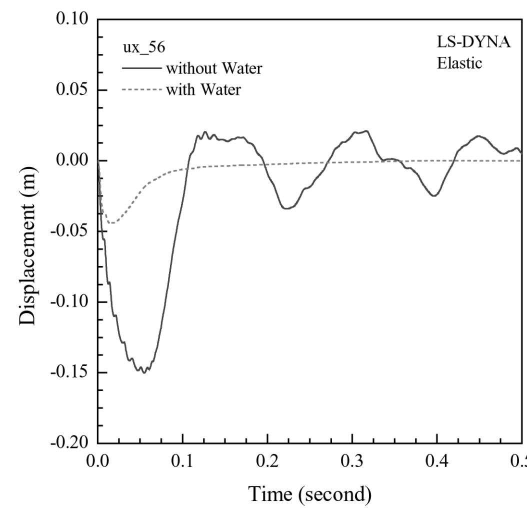

Fig. 13. Displacement Time Histories of Impact Point According to Fluid

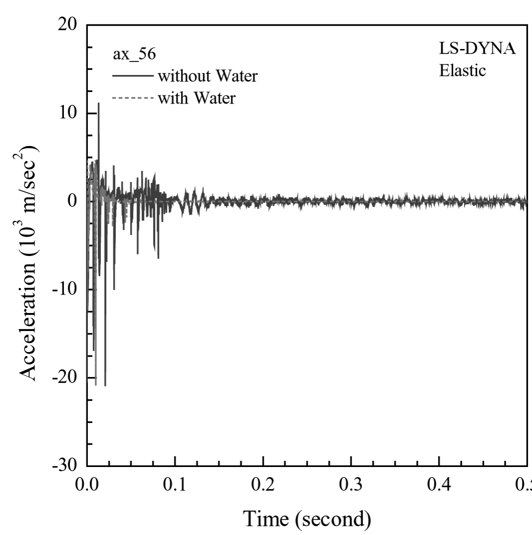

Fig. 14. Acceleration Time Histories of Impact Point According to Fluid

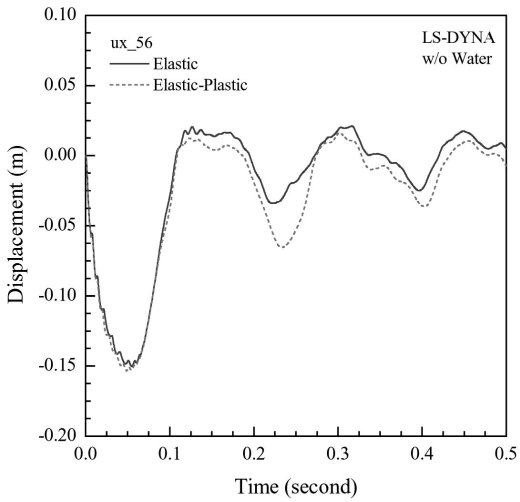

Fig. 15. Displacement Time Histories of Impact Point According to Material Properties

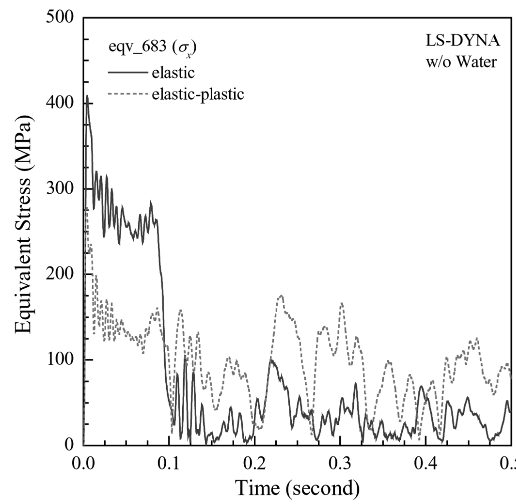

Fig. 17. Equivalent Stress Time Histories According to Material Properties

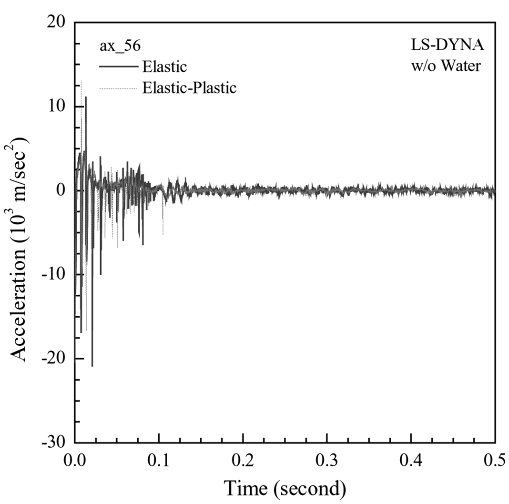

Fig. 16. Acceleration Time Histories of Impact Point According to Material Properties

attempted and their results were compared. The dynamic characteristics of the water storage tank were investigated by performing several sensitivity runs.

Based on the analyses, the following conclusions have been reached:

- Frequencies decrease by more than 50% with the inclusion of fluid in the water storage tank.
- There is little difference between the initial velocity and impact force application for the impact analysis of ANSYS.
- ANSYS and LS-DYNA generate the same response characteristics for the impact analysis.
- Fluid can be a significant factor in terms of the their capacity to absorb the response of the tank due to the impact. An impact analysis by ANSYS can be well performed for the model instead of LS-DYNA if there is no convergence problem, which is frequently encountered in real fluid-structure interaction problems.

REFERENCES

[1] UK Aircraft Accident Investigation Board, “Report on the Accident to Boeing 737-236, G-BGJL at Manchester International Airport on 22 August 1985,” Aircraft Accident Report No. 8/88(EW/C929), British Department of Transport (1985).

[2] Yu, S.O., Jo, J.C., Kim, H.J., Yune, Y.G. and Jhung, M.J., “Transient piping pressure and force caused by high pressure steam flow disturbances,” *Transactions of the Korean Society of Pressure Vessels and Piping*, Vol.1, No.1, pp.41-54 (2005).

[3] ANSYS, ANSYS *Structural Analysis Guide*, ANSYS, Inc., Houston (2004).

[4] LSTC, LS-DYNA *Theoretical Manual*, Livermore Software Technology Corporation, Livermore (1998).

[5] Grimes, R.G., Lewis, J.G. and Simon, H.D., “A Shifted Block Lanczos Algorithm for Solving Sparse Symmetric Generalized Eigenproblems,” *SIAM Journal on Matrix Analysis and Applications*, Vol.15, No.1, pp.228-272 (1994).

[6] Jhung, M.J., Jeong, K.H., Hwang, W.G., “Modal analysis of eccentric shells with fluid-filled annulus,” *Structural Engineering and Mechanic*s, Vol.14, No.1, pp.1-20 (2002).

[7] Jhung, M.J., Choi, Y.H. and Jeong, K.H., “Modal analysis of coaxial shells with fluid-filled annulus,” *Structural Engineering and Mechanics*, Vol.16, No.6, pp.655-674 (2003).

[8] Jhung, M.J., Choi, Y.H., Kim, H.J., Jeong, K.H., “Natural vibration characteristics of a clamped circular plate in contact with fluid,” *Structural Engineering and Mechanics*, Vol.21, No.2, pp.169-184 (2005).