Two-Dimensional CFD Analysis of Shock Wave Propagation for Developing Mitigation Measures of Severe Accident Explosive Loads

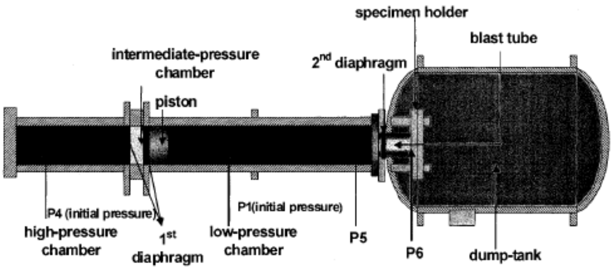
Jin-Su Kim*, Jong-Woon Park
Dongguk univ., 707, Sekjang-Dong, Gyeong Ju, South Korea
** Corresponding author: jinsur@gmail.com*

1. Introduction

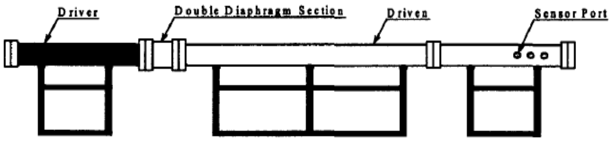
After Fukushima accident, a new need is raised for preserving integrity of vital components and structures in a nuclear power plant against blast waves from hydrogen and steam explosion during a severe accident. A research is underway for developing hydrodynamic and/or mechanical measures for mitigating such shock waves. For the research, an analysis methodology and its validation for shock wave propagation through diverse media of air, water or mixture and reflection/damping via such measures is important. As a starting point of the development, CFD method is applied for prediction of shock wave propagation in an existing shock tube experiment. And the results are discussed by comparing with previous analyses.

2. Method of Analysis

2.1 Review of Existing Shock Tube Experiments



(a) Michigan State University shock tube [1]



(b) Shock tube at Konkuk University (STKKU I [2])

Fig. 1. Shock tube experiments

Shock tubes utilize difference in pressures to generate high-enthalpy and high-speed flows. There are two typical lab shock tubes : MSU shock tube [1] made of a steel alloy containing chromium and manganese and STKKU shock tube [2] made of a stainless steel (STS 304). Both had high strength and high temperature resistance. The pictures and dimensions of the two shock tubes are shown in Fig. 1 and Table 1, respectively.

In the MSU shock tube [1], the two diaphragms separating the intermediate-pressure chamber from the high-pressure chamber and the low-pressure chamber are machined with grooves with well-calculated depth to allow the first diaphragm to burst at the pressure level equals to the high pressure. The second diaphragm bursts at the pressure equal to the difference between the high pressure and the low pressure.

Table 1 Specification of the Shock Tubes

Specification	MSU [1] (mm)	STKKU I[2] (mm)
Outer diameter of tube	120	117
Inner diameter of tube	80	93
High-pressure chamber length	2	2
Low-pressure chamber length	4	4
Blast tube length	0.1537	2

2.2 A Model for CFD Analysis

The FLUENT code is adopted to investigate the shock wave generation and propagation in two dimensions as in Fig. 2. The three-section shock tube is simulated by two sections only. The bursting process of the diaphragm is also neglected, no wall heat transfer and gas leakage are considered, and fill gas is assumed to be ideal gas of air. The influence of mesh number was studied by decreasing from 8000 cells to 4000 cells [1] and the present study used 7500 symmetric and quadrilateral cells. Viscous calculations based on laminar and Reynolds stress model (RSM), and inviscid calculations are performed to evaluate the viscous effects. 2D coupled-explicit scheme [3] is employed since its best agreement with the analytical solution was previously reported [1].

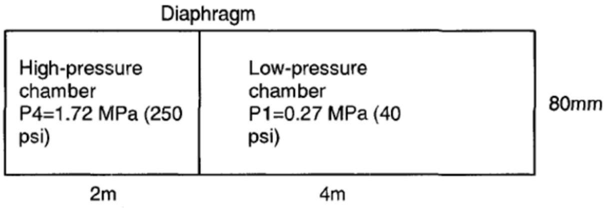


Fig. 2. A simplified Model for CFD Analysis

3. Results and Discussions

As in the experimental measurement [1], point p5 as shown in Fig. 3 is investigated. The simulation results are depicted in Figs. 4-6 and key values are summarized in Tables 2 and 3.

Overall results show nearly identical results but inviscid calculations are closest to experiment with respect to the first peak pressure and shock wave arrival time at P5. The difference of peak pressures between the experiment and calculations seems to be due to fluid losses from tube sealing, the diaphragm opening and heat loss in the experiments, which were not included in the previous and present calculations.

Viscous calculations (laminar and RSM) provide slower wave propagation speeds due to flow resistance by viscosity. While Kai Long [1] reported that the viscous effect did not play an important role for the P5 simulation, Tak, et al. [2] reported differently: their experimental shock arrival time at the right end sensor (Fig.1b) is 6 ms slower than that from viscous calculation and it is 2 μs slower than that from inviscid calculation. Present arrival times for viscous calculations were 6 ms slower than inviscid calculation, which is consistent with Tak, et al. [2].

Fig. 3. Right end point of investigation simulating sensor P5

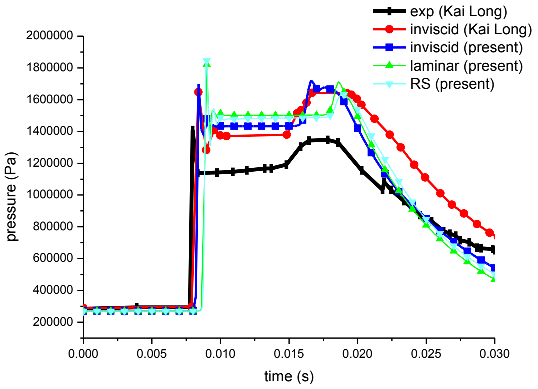


Fig. 4. Pressure at P5 from 0s to 0.03s

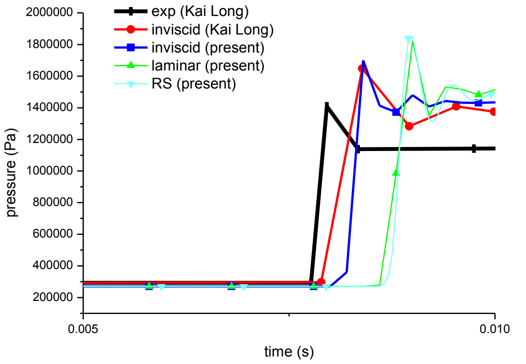


Fig. 5. Pressure at P5 from 0.005s to 0.010s

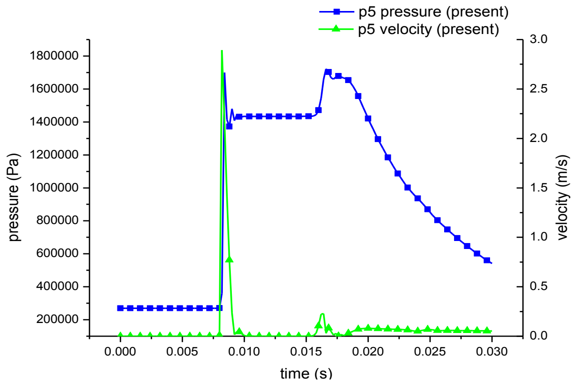


Fig. 6. Pressure and velocity at P5

Table 2 First peak pressure

Case	First peak (Mpa)
Experiment (Kai Long)	1.41
Inviscid (Kai Long)	1.65
Inviscid (present)	1.70
Laminar (present)	1.82
Reynolds stress (present)	1.84

Table 3 Shock wave arrival time at P5

Case	Arrival time (ms)
Experiment (Kai Long)	7.8
Inviscid (Kai Long)	7.9
Inviscid (present)	8.0
Laminar (present)	8.6
Reynolds stress (present)	8.6

4. Conclusions

An existing shock tube is simulated by using FLUENT for later development and modeling of hydrodynamic and/or mechanical measures for mitigating severe-accident shock waves. Viscous calculations show slower wave propagation due to flow resistance by viscosity and this is consistent with Tak, et al.'s result but not with Kai Long's result. Thus, more detailed considerations in such aspects as wall friction, heat transfer and turbulence is needed for future shock wave modeling and validation and this is underway including damping effect by shock mitigation measures.

REFERENCES

[1] K. Long, Blast Simulation with Shock Tube Testing and Computational Fluid Dynamics Analysis, Master's Thesis, Michigan State University, 2008.

[2] J. S. Tak, C. J. Huh, Y. H. Byun, and J. W. Lee, A study on the Performance Analysis of STKKU I, The Korean Society for Aeronautical & Space Sciences, Vol. 28, p. 28, 2000.

[3] K. Kato, T. Aoki1, S. Kubota, and M. Yoshida, A numerical scheme for strong blast wave driven by explosion, International Journal for Numerical Methods in Fluids, Vol. 51, p. 1335, 2006.

[4] J.Y. Kim, ANSYS CFD FLUENT, ANSYS Inc., Canonsburg, 2011.