Impact Test Simulation for Nuclear Power Plant Safety under Tornado Disaster

Sunao Tokura
Tokura Simulation Research

Abstract

In recent years, the safety standards of nuclear power plants against natural disasters, e.g., earthquake, tsunami, tornado, have been extremely intensified in Japan. Regarding the influence of tornadoes, the magnitude and the wind velocity of tornado and the geometry, dimensions, mass, impact velocity and other properties of the missiles from tornado are regulated in detail and the safety measures supposing the impact of the missile to certain vulnerable zones in nuclear power plants are required. Concerning the structures composed with steel plates in the buildings of nuclear power plants, the countermeasure to prevent the penetration of the missile to the structures should be considered. For these safety purposes, Central Research Institute of Electric Power Industry (CRIEPI) performed the free drop impact test of the impactor on steel plate which imitates the impact of tornado missile on steel structures in nuclear power plant and showed the relation between penetration criteria and impact energy[1]. On the formulation of safety standards for impact events based on experiments, the relation between strain rate dependency and triaxiality regarding failure condition of steel should be clarified. In this paper the simulation re-creating the experiment which performed by CRIEPI was tried using LS-DYNA®. Highly accurate simulation is required for the formulation of safety standards. For this purpose, strain rate dependency of the steel and the impactor are considered in the simulation and the impact forces and the displacements of both the experiment and the simulation are compared. As the result, the analysis model of LS-DYNA showed good agreement to the experiment. It is expected that the modeling technique of the drop impact test presented in this paper can be used effectively to formulate reliable safety standards of nuclear power plants.

1. Introduction

Central Research Institute of Electric Power Industry (CRIEPI) performed the free drop impact test with several types of the impactor and material, and measured the damage of the impactor, impact force, strain in the test piece (plate) and so on. The configuration of the drop test is shown in Fig.1. In this paper the rectangular pipe of the impactor and the target plate made of steel known as JIS G3101 SS400 (JIS ; Japanese Industrial Standard) were selected. SS400 is basically same steel material of ASTM A36 (ASTM ; American Society for Testing and Materials). SS400 is also the most commonly used hot rolled general structural steel. The rectangular pipe and the plate are same material, whereas the material properties are slightly different as shown in the paper of CRIEPI[1]. CRIEPI also performed numerical simulation of the drop test using hydrodynamic code AUTODYN and the results of the simulation and the experiment are compared. The purpose of this study is the creation of accurate FEM model of test equipment for LS-DYNA.

2. Analysis Model

The geometry and dimensions of the drop test model are shown in Fig.2. The test piece is a steel plate of 2,140 mm width, 1,400 mm depth and 9 mm thickness. It is fixed on the support with bolts and the stoppers. The

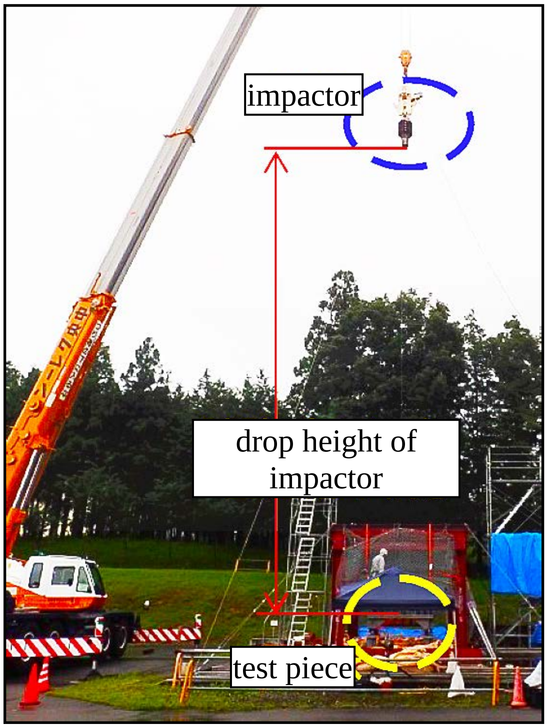


Fig.1 Drop test configuration[1]

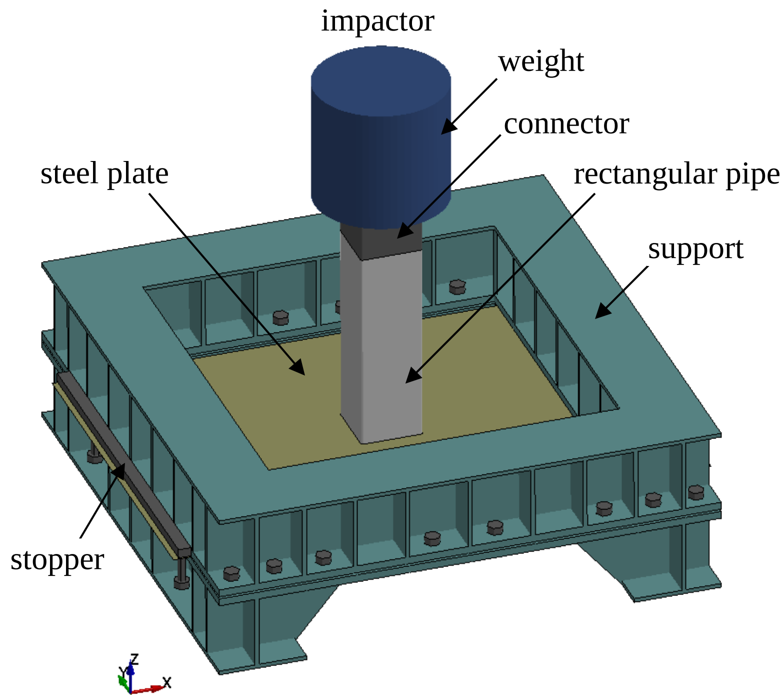


Fig.2 Geometry of analysis model

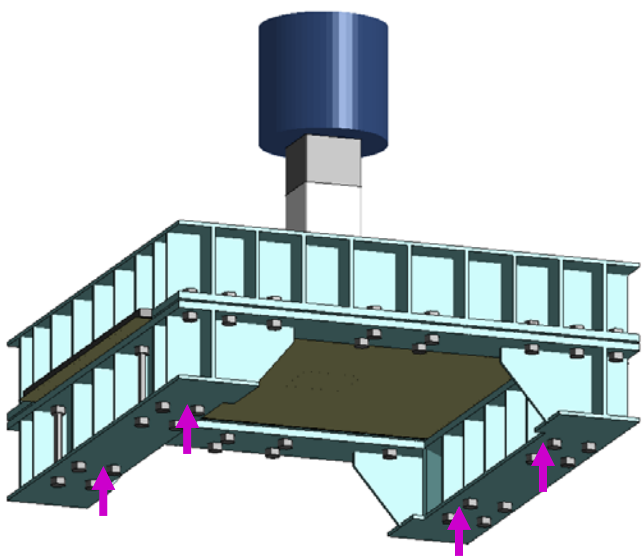


Fig.3 Location of four load cells under the support

effective width and the depth of the plate surrounded by the support are both 1,400 mm. The impactor consists of three parts, i.e., rectangular pipe, connector and weight. The dimension of the cross section of the rectangular pipe is 250 mm x 250 mm, 4.5 mm thickness, 1,000 mm length. The mass of the impactor is 1,100 kg. Four load cells are placed under the support to measure impact force. The load cells are modeled using nonlinear Maxwell spring elements.

3. Material Properties

The material properties are summarized in Table.1 and nonlinear stress-strain curves for the materials are shown in Fig.4. All the information about the materials are described on [1]. Strain rate dependency is considered based on the estimated formula of the Japan Welding Engineering Society (below WES formula). WES formula gives strain rate dependency for both yield stress and tensile strength respectively (Eq.1*a*, 1*b*).

$$\sigma_Y=\sigma_{Y0}(T_0)\cdot exp\left[8\times10^{-4}\cdot T_0\cdot\left(\frac{\sigma_{Y0}(T_0)}{E}\right)^{-1.5}\cdot\left\{\frac{1}{T\cdot ln\left(\frac{10^8}{\dot{\varepsilon}}\right)}-\frac{1}{T_0\cdot ln(\frac{10^8}{\dot{\varepsilon}_0})}\right\}\right] \quad (1a)$$

$$\sigma_T=\sigma_{T0}(T_0)\cdot exp\left[8\times10^{-4}\cdot T_0\cdot\left(\frac{\sigma_{T0}(T_0)}{E}\right)^{-1.5}\cdot\left\{\frac{1}{T\cdot ln\left(\frac{10^9}{\dot{\varepsilon}}\right)}-\frac{1}{T_0\cdot ln(\frac{10^9}{\dot{\varepsilon}_0})}\right\}\right] \quad (1b)$$

Where, σ_Y ; yield stress, σ_T ; tensile strength, $\dot{\varepsilon}$; strain rate, T ; temperature, and E ; Young's modulus. The subscript 0 denotes the baseline of these values. As only one definition of strain rate dependency for one nonlinear stress-strain curve is given in LS-DYNA, two rate dependency curves of WES formula are averaged and the average curve was fitted using Cooper-Symonds rate dependency model. The parameters C and p of Cooper-Symonds model were identified to meet the average curve using LS-OPT® (Fig.5). The load cells are modeled using Maxwell nonlinear spring. The parameters K_0, K_i and β of Maxwell model were also identified using LS-OPT.

Table.1 Material properties

Part	Material type	Density (ton/mm^3)	Young's modulus (N/mm^2)	Poisson's ratio	Yield stress (N/mm^2)	Cooper-Symonds parameter	
						C	p
Plate	*MAT_PIECEWISE_LINEAR PLASTICITY	7.86x10^{-9}	209700	0.3	322.3	2536.99	5.30668
Pipe		7.86x10^{-9}	205800	0.3	287.5	882.853	5.12353
Connector		7.86x10^{-9}	205800	0.3	287.5	882.853	5.12353
Weight	*MAT_RIGID	8.4208x10^{-7}	205800	0.3	-	-	-
Support		9.432x10^{-9}	205000	0.3	-	-	-
Bolts		9.432x10^{-9}	205000	0.3	-	-	-
Stopper		9.432x10^{-9}	205000	0.3	-	-	-

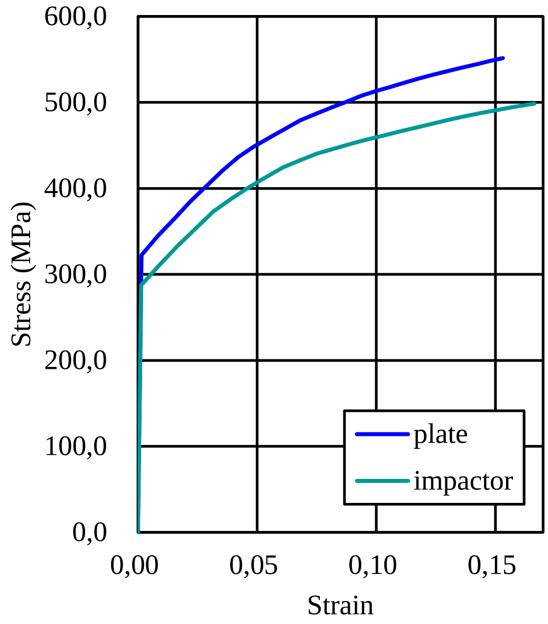


Fig.4 Stress-strain curve for target plate and rectangular pipe of impactor

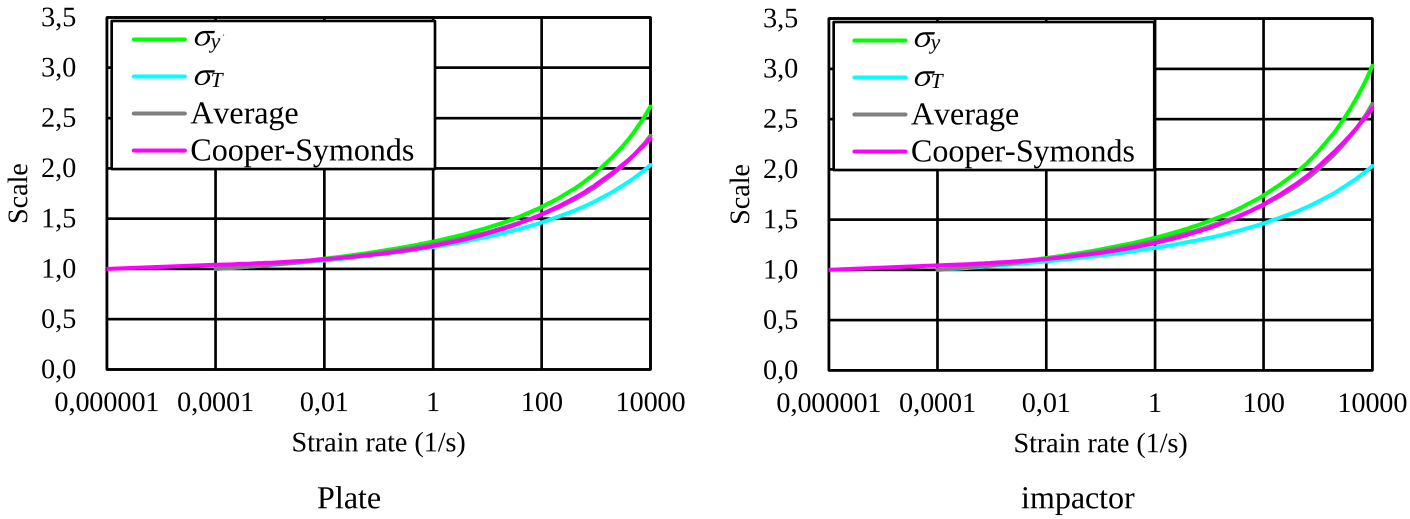


Fig.5 Strain rate dependency curves from WES formula are fitted using Cooper-Symonds curves

4. Analysis Conditions

The impact velocity is 18.26 m/s since the impactor drops from the height of 17 m. The contacts in the model are defined as follows;

Impactor – Plate : *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE
Rectangular pipe : *CONTACT_AUTOMATIC_SINGLE_SURFACE
Plate – Stopper : *CONTACT_TIED_NODES_TO_SURFCAE_OFFSET
Plate – Support : *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

Gravity load is also considered to include initial deflection of the plate.

5. Results

The deformation and the plastic strain contour is shown in Fig.6. Spring back occurred at 0.0395 seconds. The deformed shapes of the rectangular pipe from the experiment and the simulation are shown in Fig.7. Deformation mode of the simulation is different from that of the experiment. Deformation of the plate of the experiment and the deformation and the thickness reduction from the simulation are compared in Fig.8. It seems that the impactor contacts to the plate with slight inclination. The maximum thickness reduction in the paper[1] is 1.2 %, while it is 12.8 % in the simulation. The displacement history of the plate from the experiment and the simulation are compared in Fig.9. The displacement on the plate is measured at three points, i.e., the center of the plate and two points at -150 mm and 150 mm apart from the center respectively. Even these two points are located symmetrically, the displacement history of these two points from the experiment are different. The paper[1] also compares the impact force of the experiment with the simulation result of AUTODYN. These force curves are also compared with the result of LS-DYNA shown as Fig.10.

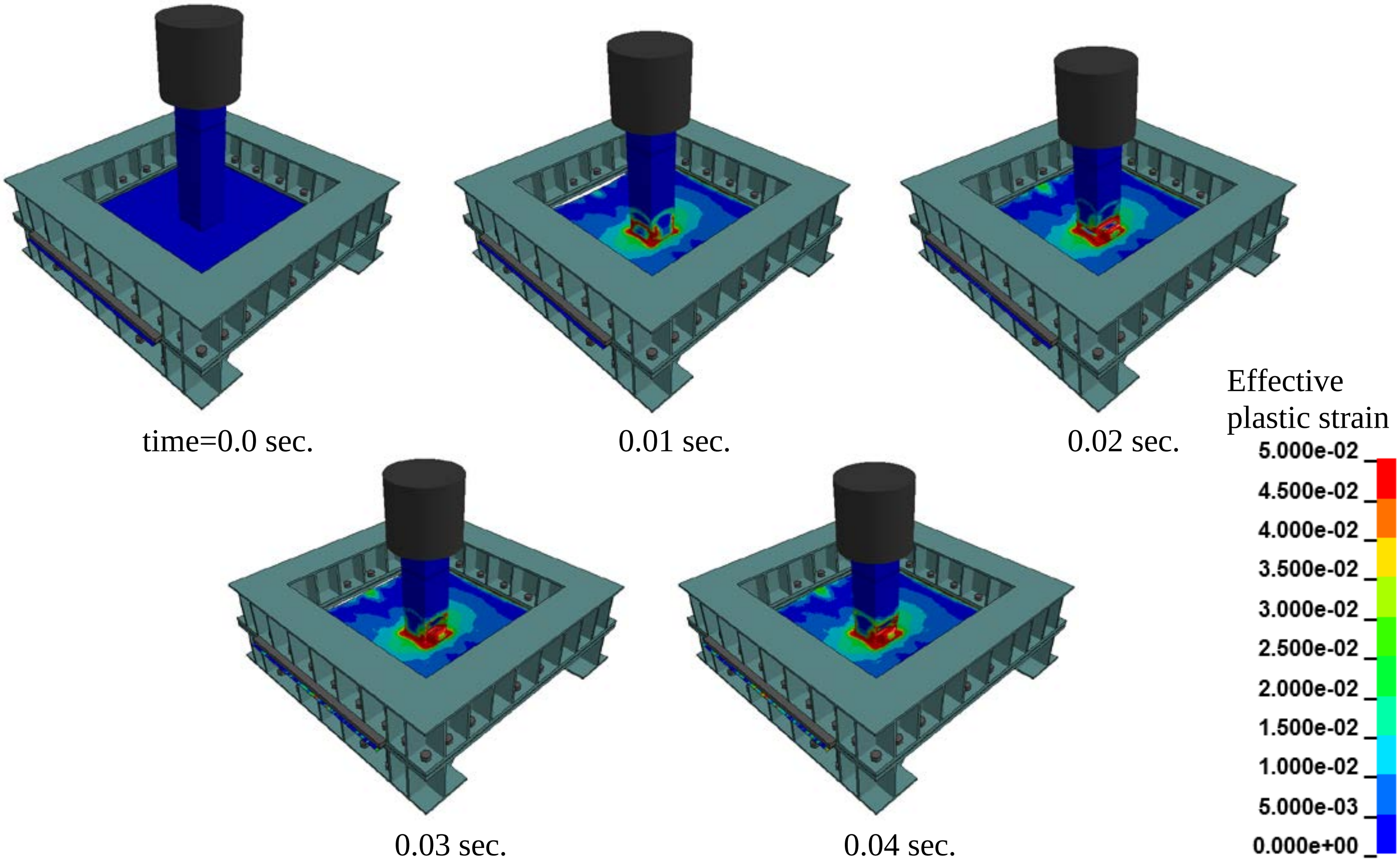
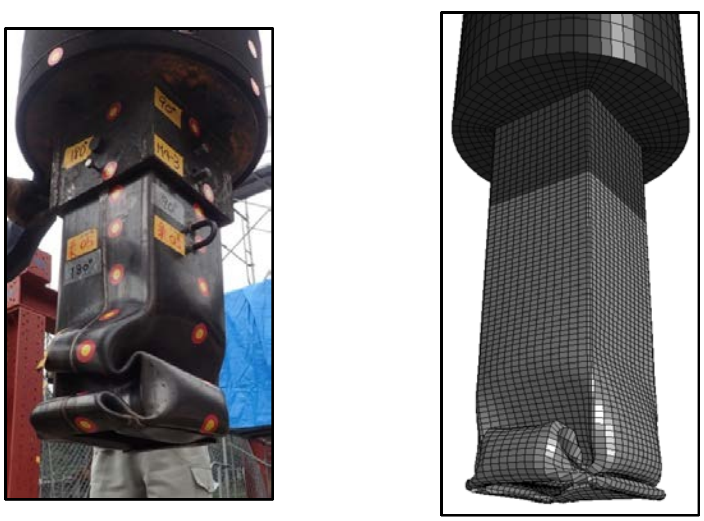
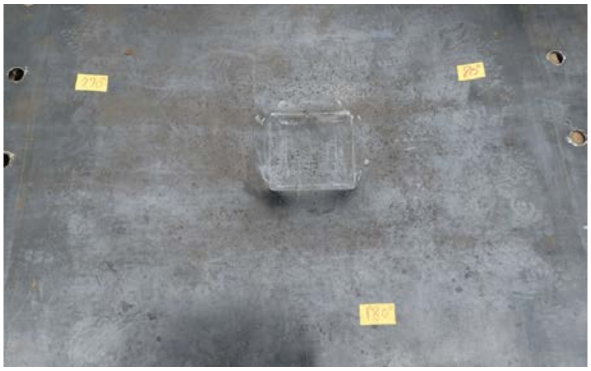


Fig.6 Deformation and effective plastic strain contour of the simulation using LS-DYNA

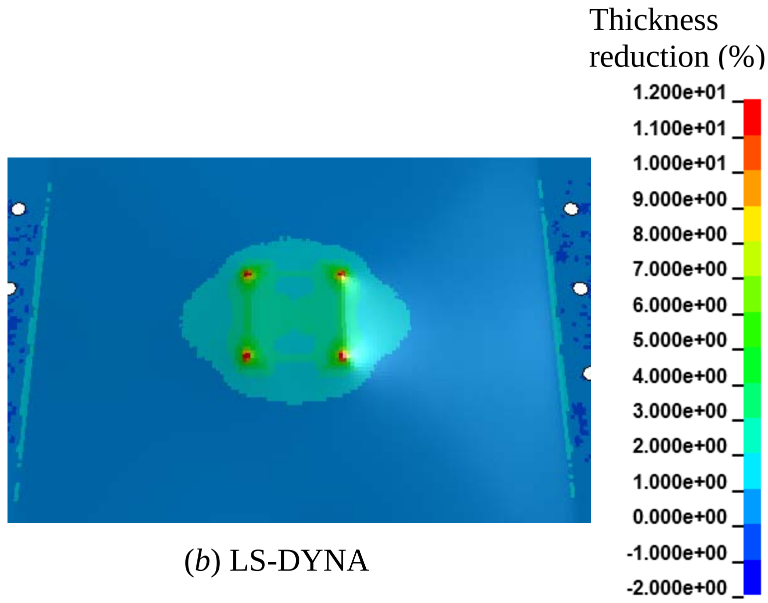


(*a*) Experiment (*b*) LS-DYNA

Fig.7 Comparison of deformation of rectangular pipe

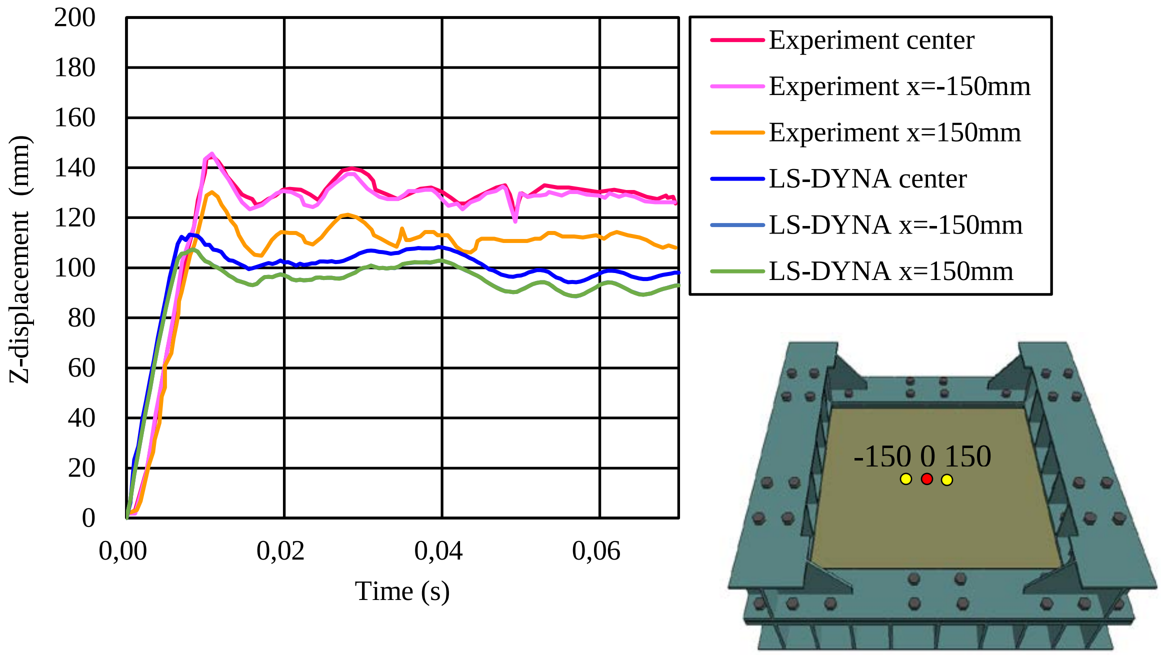


(*a*) Experiment



(*b*) LS-DYNA

Fig.8 Deformation of plate from the experiment and thickness reduction contour from the simulation



Measurement location

Fig.9 Displacement history of plate

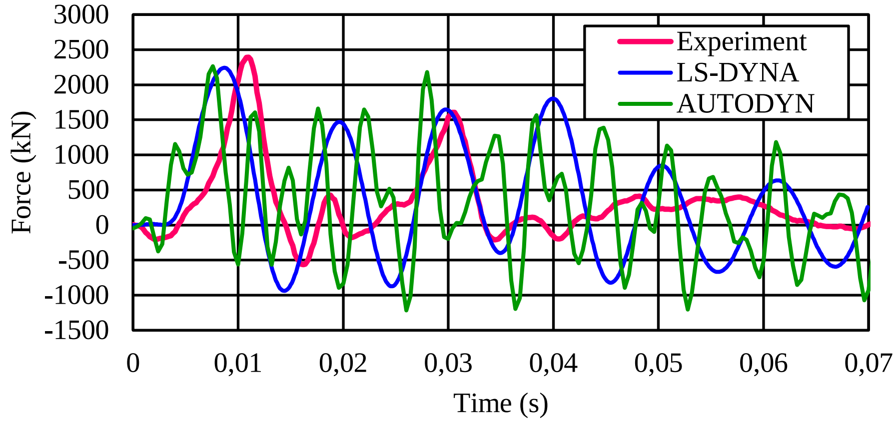


Fig.10 Impact force history

6. Conclusion and future work

The simulation of the experiment to establish the safety standard of nuclear power plant against tornado missile was tried using LS-DYNA. The experiment was a drop test of the impactor with a rectangular pipe on the steel plate. Very accurate modeling technique including reliable material properties, strain rate dependency, and so on is required to obtain good agreement to the experiment. In this trial, however, several disagreement can be seen. The thickness reduction and the displacement of the plate from the simulation are different. And the deformation mode of the rectangular pipe is also different. Further investigation on the analysis condition, the experimental condition, and measurement procedure at the experiment are required.

References

[1] Sakamoto Y., Shirai K., Udagawa T., and Kondo S., Proposal of Evaluation Methodology to Estimate Penetration Thickness of Steel Plate due to the Tornade Missile Impact by Free Drop Impact Test of Heavy Weight with Rectangular Pipe Node Shape, N15004, Central Research Institute of Electric Power Industry, 2015 (in Japanese)

[2] Japan Nuclear Technology Institute, Design Guideline to Prevent the Pipe Rupture Accident due to Combustion of Radiolysis Gas in BWR Steam Piping, JANTI-NCG-01-Third edition, 2012 (in Japanese)