



## Development of Hardware-in-the-Loop Simulator for Piping Integrity Evaluation

Y. J. Kim<sup>1)</sup>, J. B. Choi<sup>1)</sup>, N. S. Huh<sup>1)</sup>, C. Y. Nam<sup>1)</sup> and C. R. Pyo<sup>2)</sup>

1) *Sung Kyun Kwan University, Korea*

2) *Induk Institute of Technology, Korea*

### ABSTRACT

In order to verify the analytical methods predicting failure behavior of cracked pipes, full-scale pipe tests are crucial in nuclear power plant pipings. For this reason, series of international test programs have been conducted. However, full-scale pipe tests require expensive testing equipment and long period of testing time. The objective of this paper is to develop a test system that can economically simulate the full-scale pipe test regarding the integrity evaluation. This system provides the failure behavior of cracked pipe by testing a wide-plate specimen. The system was developed for the integrity evaluation of nuclear pipings based on the methodology of hardware-in-the-loop (HiL) simulation. Using this simulator, piping integrity evaluation can be performed based on elastic-plastic behavior of full-scale pipe, and the high cost full-scale pipe test may be replaced with this economical system.

### 1. INTRODUCTION

Recently, the leak-before-break (LBB) concept was accepted as a technically justifiable approach for piping design of new plant and operating plant. In this case, the evaluation methods and analysis procedures must be verified. The increase in number of cracks found during the service has raised the necessity of developing more sophisticated analytical and experimental methods for the application of LBB concept.

Applying LBB design concept to piping system, the material properties, such as stress-strain relation and fracture resistance curve are required for elastic-plastic fracture mechanics analysis. The fracture mechanics analysis based on  $J$ -integral and tearing modulus has been successfully adopted for the application of LBB in standard pipings. Recently, issues on dynamic strain aging and reverse cyclic loading have been raised regarding the application of LBB. In order to investigate these complicated problems, alternative analytical methods are required and experimental verifications should also be followed. For this purpose, series of international test programs[1-4] have been conducted. However, limited number of full-scale pipe tests was carried out due to the high cost equipment and long period of testing time.

In the present paper, the methodology of hardware-in-the-loop (HiL) simulation has been introduced to replace the high cost time consuming full-scale pipe tests. This approach utilizes a software algorithm for simulating full-scale structure behavior, and the output of this simulation is interfaced to real hardware components such as wide-plate specimens mounted on a test equipment. The methodology of HiL was originated in the aerospace and

the defense industries where it is frequently impossible, impractical, or just too costly to test controllers on actual systems, and popularly used in automobile industries in order to improve the component performance[5-7]. Suh and Kim[8] developed a real time simulator for an anti-lock brake system (ABS) based on the methodology of HiL simulation using a personal computer. A. Ptak and K. Foundy[9] developed a real time simulator for the verification and validation testing of an attitude control system for spacecraft.

The objective of this paper is to develop a test system which can economically simulate the full-scale pipe test regarding the integrity evaluation based on the methodology of HiL simulation.

## 2. HiL SIMULATOR FOR PIPE INTEGRITY EVALUATION

The proposed HiL based piping integrity evaluation simulator consists of software part, hardware part and input/output (I/O) part as shown in Fig. 1. The software part is composed of elastic and elastic-plastic finite element analysis module. The hardware part is composed of test machine, wide-plate specimen, sensors, and display unit. The I/O part consists of A/D and D/A converters for the data transmission between software and hardware parts.

### 2.1 Software Part

#### - Loading schedule module

The loading schedule module provides information on the variation of loading conditions to the finite element model. The finite element model thus deforms according to the actual loading conditions which are observed from the corresponding nuclear piping. In this paper, a remote bending condition was considered.

#### - Finite element analysis module

In this module, series of elastic finite element analyses are conducted on a full-scale pipe to obtain the boundary conditions of a wide-plate specimen which is actually mounted on the testing machine. For the finite element analysis, the commercial finite element analysis program ABAQUS[10] was used. The resulting boundary conditions for the wide-plate specimen are transferred to the main control PC as shown in Fig. 1.

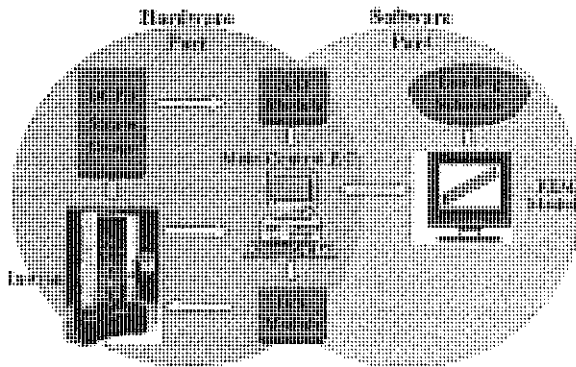


Fig. 1 A schematic illustration of piping integrity evaluation simulator.

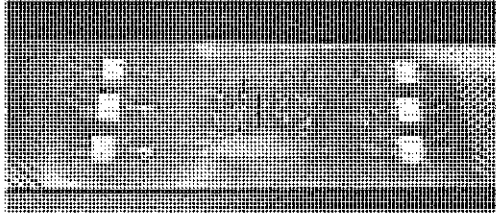


Fig. 2 Wide-plate specimen.

## 2.2 Hardware Part

### - *Specimen and test machine*

Fig. 2 shows the wide-plate specimen with through-wall crack which were sharpened with fatigue pre-crack at both sides. Six strain gages were mounted on the surface to produce the equivalent boundary conditions which were obtained from the finite element analysis for the full-scale pipe, as shown in Fig. 2. An Instron testing machine (model 8503) was used for applying the loading condition to the wide-plate specimen and was automatically controlled by an electrical motor. Uni-axial tension was applied to the specimen according to the loading schedule produced by the software part.

### - *Strain measurement system*

A specially designed strain measurement system was applied to match the strain field in the wide-plate specimen with the finite element analysis. In order to minimize the measurement error during the loading process, total of 6 strain gages were attached to the specimen as shown in Fig. 2. A multi-channel strain measurement system covering 16 channels was developed for this purpose. The multi-channel strain measurement system consists of wheatstone bridge, channel control part, filter, amplifier, and A/D converter. The signals from the attached strain gages are transferred into the wheatstone bridge, the filter, the amplifier, the A/D converter and finally recorded in the main control PC. The developed low-pass filter has a cut-off frequency of 10Hz and the developed amplifier has a gain of 40dB, 54dB, 60dB, and 74dB.

### - *Crack detection system*

The crack growth behavior is monitored by a multi-channel DCPD (Direct Current Potential Drop) system. DCPD is an indirect method of measuring crack sizes. It provides the amount of crack growth in terms of voltage drop. In this paper, three DCPD channels were used for voltage reading, as shown in Fig. 2. The monitored data are recorded in the main control PC.

In order to determine the crack length from voltage change, the following equation which was analytically developed by Johnson[11] was used.

$$\frac{V}{V_0} = \frac{\cosh^{-1}[\cosh(\pi y / 2W) / \cos(\pi a / 2W)]}{\cosh^{-1}[\cosh(\pi y / 2W) / \cos(\pi a_0 / 2W)]} \quad (1)$$

Eq. (1) is valid for all values of  $a/W$ . This equation can be rewritten in terms of voltage for calculating crack length as following;

$$a = \frac{2W}{\pi} \cos^{-1} \frac{\cosh(\pi y / 2W)}{\cosh\left\{ (V / V_o) \cosh^{-1} [\cosh(\pi y / 2W) / \cos(\pi a_o / 2W)] \right\}} \quad (2)$$

where  $a_o$  and  $V_o$  are the initial crack length and the corresponding initial potential, respectively, and  $a$  and  $V$  are the current crack length and the corresponding potential, respectively.

Fig. 3 illustrates the wide-plate specimen geometry and wire placement locations for this solution.

- *A/D converters*

The analog voltage measurement obtained from the DCPD system is converted to digital signal by passing the A/D converter which was specially designed for the developed DCPD system. The resulting data are graphically displayed on the main control PC to monitor the crack growth behavior.

- *Main Control PC*

Main control PC is in charge of data communication between hardware and software parts, data storage, and loading schedule control. It also provides the graphical crack growth monitoring display in accordance with the DCPD measurement. A high-performance personal computer (Pentium II 350 MHz) was used for this purpose.

### 3. PROCEDURES OF HiL SIMULATION

Procedure of HiL simulation for the piping integrity evaluation is explained as follows.

- 1) In the first stage, the finite element analysis is conducted for full-scale pipe model in accordance with pipe loading schedule. As a result, a loading schedule for the wide-plate specimen is produced in terms of strain.
- 2) The wide-plate specimen is mounted on the testing machine with sensors connected. The strain measurement system and the DCPD system are initialized.
- 3) The resulting boundary conditions for the wide-plate specimen is transmitted to the main control PC. The main control PC then initiates the testing machine by controlling loading motor.
- 4) The measured strain values due to the loading are transmitted to the main control PC, and are compared with the strain value which was set by the loading schedule. The applied load increases until the measured strain matches with that of loading schedule.
- 5) At the same time, the crack growth behavior is monitored by the DCPD system. The amount of crack growth is recorded as a function of strain which is specified in the loading schedule.
- 6) The above procedures are repeated until the loading schedule is completed.

### 4. APPLICATION OF HiL SIMULATOR

#### 4.1 Determination of Specimen Size

In order to determine the wide-plate specimen size, a finite element analysis for the full-scale pipe model was performed. Two planes of symmetry were considered in modeling a full-scale pipe, and thus the designed finite element model represents only a quarter of pipe as shown in Fig. 4.

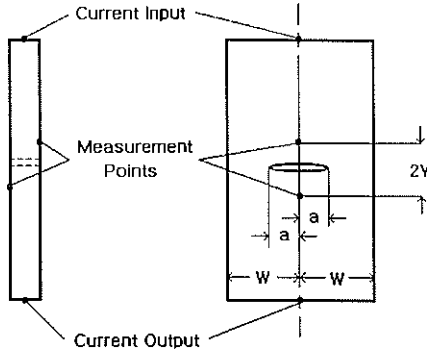


Fig. 3 Wide-plate specimen geometry and electric potential wire placement locations for Johnson's formula.

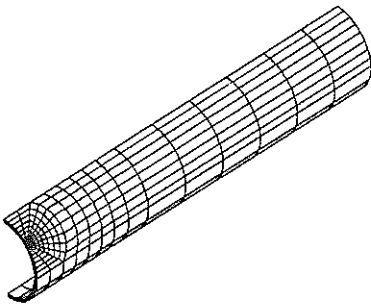


Fig. 4 A three dimensional mesh design for the finite element analysis.

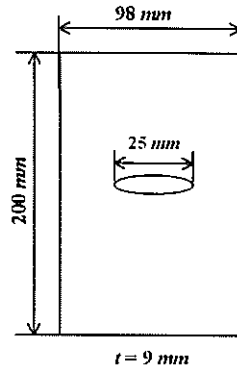


Fig. 5 The configurations of wide-plate specimen.

Table 1 Material properties for the wide-plate specimen.

Wide-plate specimen (SM45C)	Young's modulus, E (GPa)	207
	Yield strength, $\sigma_y$ (MPa)	466
	Ultimate strength, $\sigma_u$ (MPa)	977
	Poisson's ratio, $\nu$	0.3

From the finite element analysis result, the size of wide-plate specimen was determined considering the strain field which was not influenced by the local crack tip strain field. For the sake of testing simplicity, a flat wide-plate was used for the test. Fig. 5 shows the configurations of wide-plate specimen for the HiL simulation. Table 1 shows the material properties of wide-plate specimen used for the HiL simulation.

## 4.2 HiL Simulation with Wide-Plate Specimen

For the HiL simulation, a remote bending moment of 200  $kN\cdot m$  to the full-scale pipe was considered. According to the resulting finite element analysis, the loading schedule was set by 15 steps in terms of strain. A wide-plate specimen was tested with initial crack size of 22  $mm$ . During the simulation, the strain values obtained from the strain measurement system were recorded along with the corresponding crack sizes at each loading step.

Fig. 6 shows the comparisons between the strain values from the measurement system and the loading schedule. The strain values measured at each loading step are indicated by solid square symbols in Fig. 6. The measured strain values showed a good agreement with those specified in the loading schedule. This implies that the motor driven loading control system operated well according to the loading schedule, and it can be utilized to correlate the finite element analysis results to the wide-plate specimen.

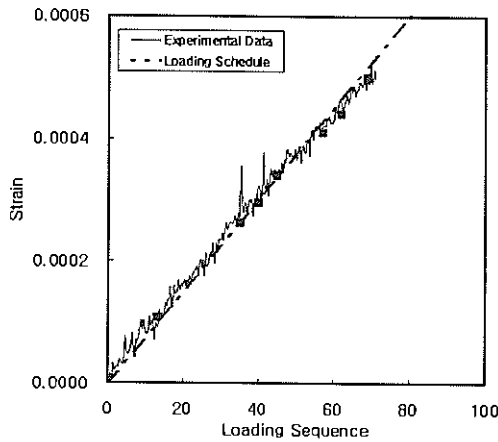


Fig. 6 Comparison of strain values between loading schedule and experimental result.

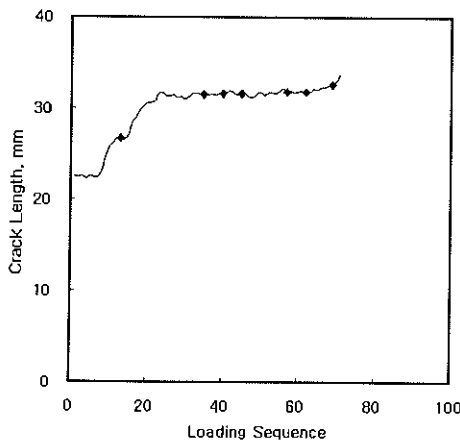


Fig. 7 Variation of crack length during the simulation at each loading step.

Fig. 7 shows the measured crack growth during the simulation at each loading step. By measuring the crack growth simultaneously at each loading step, the crack growth behavior was monitored with the increase in load applied to the full-scale pipe. The wide-plate specimen failed after seven loading steps. The final crack size measured from the DCPD system showed a good agreement with the measurement obtained from the failed specimen with less than 1% of difference, and thus the validity of applied DCPD system was proven to be reliable.

## 5. CONCLUSION

Recently, series of full-scale pipe tests were performed to verify the analytical methods predicting failure behavior of cracked pipes. In this paper, a test system which can economically simulate the full-scale pipe test regarding the piping integrity evaluation was developed. The system was developed for the integrity evaluation of nuclear pipings based on the methodology of hardware-in-the-loop (HiL) simulation. By simulating a full-scale pipe under remote bending, each part of HiL simulator, such as software part, motor driven loading control system, strain measurement system, and DCPD system was tested. This HiL simulator is expected to be utilized to verify the crack initiation load and maximum load which are currently predicted by engineering  $J$ -estimation scheme such as GE/EPRI method, limit load method et al. By modifying the developed simulator, more complicated pipeline failure behavior, such as dynamic loading and reverse cyclic loading (RCL), are also expected to be simulated. Using this simulator, piping integrity evaluation can be performed based on elastic-plastic behavior of full-scale pipe, and the high cost full-scale pipe test may be replaced with this economical system.

## ACKNOWLEDGEMENT

The authors are grateful for the support provided by a grant from the Ministry of Science and Technology (MOST) and Safety and Structural Integrity Research Center at the Sungkyunkwan University.

## REFERENCES

- [1] Wilkowski, G.M. et al., "Degraded Piping Program – Phase II," *NUREG/CR-4082*, Vol. 1 – Vol. 8, 1984
- [2] Wilkowski, G.M. et al., "Short Cracks in Piping and Piping Welds," *NUREG/CR-4599*, Vol. 1, No. 1 – Vol. 4, No. 1, 1994
- [3] Wilkowski, G.M. et al., "International Piping Integrity Research Group Program," Final Report, 1991
- [4] Wilkowski, G.M. et al., "The Secondary International Piping Integrity Research Group (IPIRG-2) Program – Final Report," *NUREG/CR-6452*, 1996
- [5] Kempf, D., Bonderson, L. and Slafer, L., "Real Time Simulation for Application to ABS Development," *SAE Paper 870336*, Society of Automotive Engineers Inc., U.S.A., 1987
- [6] Michales, L., "The Use of a Graphical Modeling Environment for Real Time Hardware-in-the-Loop Simulation of Automotive ABS System," *SAE Paper 930507*, Society of Automotive Engineers Inc., U.S.A., 1993
- [7] Sailer, U. and Essers, U., "Real Time Simulation of Trucks for Hardware-in-the-Loop Application," *SAE Paper 942297*, Society of Automotive Engineers Inc., U.S.A., 1994
- [8] Suh, M.W. and Kim, Y.J. et al., "Hardware-in-the-Loop Simulation for ABS," *SAE*

980244, Society of Automotive Engineers Inc., U.S.A., 1998

- [9] Ptak, A. and Foundy, K., "Real-Time Spacecraft and Hardware-in-the-Loop Testing," *Proceedings of 4<sup>th</sup> Real-Time Technology and Applications*, pp. 230-235, 1998
- [10] ABAQUS User's manual, Hibbitt, Karlsson & Sorensen, Inc., 1998
- [11] Johnson, H.H., "Calibrating the Electric Potential Method for Studying Slow Crack Growth," *Materials Research and Standards*, Vol. 5, No. 9, pp. 442-445, 1965