Transactions of the Korean Nuclear Society Autumn Meeting
Changwon, Korea, October 20-21, 2022

A Study on seismic analysis according to the methodology for considering experimental dynamic characteristics

ChiWoong Ra [a], Eun-ho Lee [a], Hyungyu Roh [a], and No-cheol Park [a*]
[a]*Department of Mechanical Engineering, Yonsei Univ., 50, Yonsei-ro, Seoul, 03722, Republic of Korea*
[*]*Corresponding author: pnch@yonsei.ac.kr*

1. Introduction

Under beyond design basis earthquake, nonlinear behavior due to plastic deformation is not neglectable [1]. This is a noticeable in the nuclear power plant (NPP) piping system, and therefore, research on the prediction using FE analysis of fatigue cracks in the piping system is active [2-5]. Our research team is trying to construct an FE model that can represent the acceleration and strain responses of the piping system experiment carried by Chosun University in 2022. As one of the strategies constructing FE model, we classified methods to determine the dynamic characteristics with the experiment. Furthermore, we compared the difference in seismic response when applying each method to the elastic and inelastic seismic analysis.

2. Experimental Design

As shown in Fig 1, the target piping system composed of 7 elbows is placed, and two jigs are installed at both ends of the pipe. The shaking table on the left side of Fig 1 is in a fixed state, and the shaking table on the right side is only excited with PGA 0.28g and 1.2g to the forward and backward direction. As an input for the analysis, the displacement attached to the adjacent point where the piping system and the zig were combined was used. The pipe specimen is 4-inch sch40, the straight pipes are made of SA312 TP316, and the elbows are made of SA403 WP316. In addition, 400kg and 300kg weights were attached at two locations to improve dynamic response as well as accelerate damage on elbow specimen.

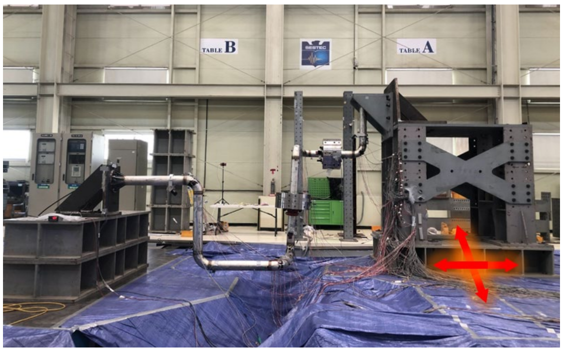


Fig. 1. Experimental set up for shaking table test consisting stainless steel specimen.

3. FE Model for Analysis

In FE analysis, the structural stiffness of the system which affect the intrinsic frequency of the entire system is mainly determined by the material properties, and the boundary conditions. Therefore, in this study, an analysis was attempted on the effect of constructing an FE model through tuning of the modulus of material and the boundary conditions on the seismic responses. Adding to that, all analysis cases were carried out through 4% damping including inelastic analysis. In addition, the experimental data were sampled at 512 Hz, and the input in the analysis was resampled to 200 Hz with the low pass filter of 50 Hz applied.

3.1 Modification of Elastic Modulus

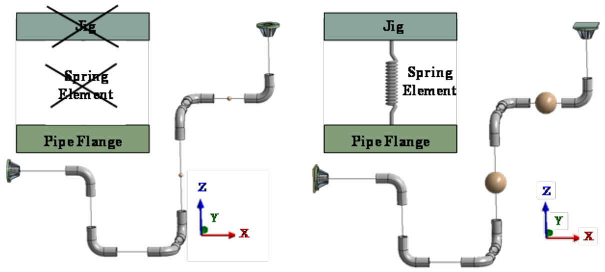
As shown in Fig 2 (a), an FE model was constructed only for piping including flange. Previously, it was assumed that the coupling between the pipe and the jig was very strong at the base of the construction methodology, so it was assumed that the same behavior appeared between the jig and the flange. Therefore, the displacement value attached to the zig was input to the top surface of the flange.

At this time, the rigidity of the whole system was changed by changing the elastic modulus of the material, and through this, the error with the natural frequency obtained in the experiment was reduced to within 0.5%. At this time, the change range of Young's modulus was limited to within 10%.

3.2 Spring Element on Boundary Section

Among the known inelastic seismic analysis on piping system, the SEGP-1-2200 presented in JSME CC suggests that the stiffness of the supporting structure, such as jig and support should be considered [5]. Based on this approach, it was confirmed that the error with the experimental natural frequency could be reduced by adjusting the adhesion to the jig and the rigidity of the jig itself.

As illustrated in Fig. 2(b), we focused on the stiffness tuning between the flange top surface and the point measuring displacement in the support structure. For the corresponding coupling area, a linear spring element was attached under assuming an elastic behavior between them. In addition, the spring element reduced the experimental natural frequency and error to within



(a) (b)

Fig. 2. FE analysis model by (a) tuning Young's modulus of material specimens, (b) applying 6-DOF spring element.

0.5% by changing the stiffness value of 6 degrees of freedom. Specifically, not only the control of the existing for the translational DOF but also the degree of constraint for the rotational DOF were simultaneously considered.

4. Analysis Results

Before proceeding with the inelastic analysis, elastic analysis was performed to confirm the consistency between the two approaches.

The time history acceleration and von-Mises stress were compared to compare the results. The vulnerable point derived from the analysis result corresponds to a specific position between the intrados and the crown, but for comparison with the attached accelerometer in the experimental environment, we compared results on the elbow crown. In addition, the circumferential strain was compared, which is dominant in cracks generated in the axial direction of the crown, which is the failure mode of thin pipe like the specimen.

The actual experimental excitation levels were performed at PGA 0.28G and 1.2G, and the input level for comparison with the experiment of seismic analysis was limited to PGA 1.2g. Considering that the time at which the maximum stress occurs is the same and the difference between the maximum values shows an error of less than 1%, and the other results listed on Table I. the largest difference on results between two model is 3.0 %, so we conclude that the elastic response between the two models is consistent.

Table I: Differences on elastic analysis results

Model / Result	Young's modulus	6 DOF spring stiffness
Von-Mises stress	-	0.5 %
Acceleration		2.7 %
Circumferential Strain		0.8 %

5. Conclusion

For the experimental system in which the relative seismic displacement between supports occurred, seismic analysis was performed by two different approaches considering experimental dynamic characteristics. As a results, the consistency of each approach was verified in the elastic behavior. Specifically, using a 6-DOF spring element, a FE model similar to the experimental dynamic characteristics was constructed by quantifying the stiffness between the measurement point and the end surface on piping system, which is included in the target system. Furthermore, it is necessary to confirm the applicability of the method through application in the later inelastic analysis.

REFERENCES

[1] I. NAKAMURA. Approaches to establish the seismic safety evaluation procedure based on the inelastic response analyses. 2017.

[2] J.B. Park. Seismic responses of nuclear reactor vessel internals considering coolant flow under operating conditions. Nuclear Engineering and Technology., 51.6, 2019, pp. 1658-1668.

[3] A.R. Kiran, G. R. Reddy, M. K. Agrawal, Seismic fragility analysis of pressurized piping systems considering ratcheting: A case study. International Journal of Pressure Vessels and Piping, 169, 2019, pp. 26-36.

[4] J.S. Kim, J.Y. Kim. Simplified elastic-plastic analysis procedure for strain-based fatigue assessment of nuclear safety class 1 components under severe seismic loads. Nuclear Engineering and Technology, 52.12, 2020, pp. 2918-2927.

[5] J.Y. Kim, J.M. Lee, J.G. Park, J.S. Kim, M.K. Cho, S.W. Ahn, G.H. Koo, B.H. Lee, N.S. Huh, Y.J. Kim, J.I. Kim, I.K. Nam, Round robin analysis to investigate sensitivity of analysis results to finite element elastic-plastic analysis variables for nuclear safety class 1 components under severe seismic load, Nuclear Engineering and Technology. 54, 2022, pp. 343-356.