Finite Element Modeling of Control Element Drive Mechanism for Elastic-Plastic Analysis

Won Ho Lee [a*], Jin Seok Park [a], Jong Min Kim [a]
[a]*KEPCO Engineering & Construction, Inc., Yuseong-gu, Daejeon, Korea*
[*]*Corresponding author: wonho13@kepco-enc.com*

1. Introduction

A control element drive mechanism (CEDM) is a safety related component as it controls the reactivity of the reactor and retains the pressure of the reactor coolant. Therefore, the CEDM shall be designed to maintain its structural integrity and functions under design basis earthquake (DBE) [1]. After the Fukushima nuclear accident, a beyond-design basis earthquake (BDBE) has been a new consideration for nuclear component design. Conventional structural finite element (FE) models for the CEDM mainly consist of equivalent beam and mass elements for linear elastic analysis and it has been sufficient to use simplified elastic-plastic analysis [2] in case that the results of the elastic analysis did not meet the acceptance criteria. However, nonlinear elastic-plastic analysis is required to incorporate the BDBE due to its high level. This paper presents the procedure for development of the FE model of the CEDM for nonlinear elastic-plastic analysis including sensitivity and convergence study.

2. Methods and Results

2.1 Target Model

The CEDM consists of the pressure boundary components and the non-pressure boundary components as shown in Fig. 1. The pressure boundary components of the CEDM are the upper pressure housing (UPH), the motor housing (MH) assembly and the reactor vessel closure head nozzle. The other components including extension shaft assembly (ESA), the motor assembly, the upper shroud (US) assembly and the reed switch position transmitter (RSPT) are the non-pressure boundary components. Table 1 lists the materials of the pressure boundary components and the US.

2.2 Material Model

The material properties of the components in Table 1 including modulus of elasticity and poison's ratio at design temperature, 343°C, were taken from [2] and [3]. The Chaboche kinematic hardening model combined with the Voce isotropic hardening model was used except that the bilinear kinematic hardening model was applied to the MH. The combined Chaboche model parameters were collected from [4], [5], [6], [7] and those of bilinear model were calculated by fitting the true stress-strain curve of the 410 stainless steel from [8] and using the elasticity of modulus and yield strength from [3] as shown in Table 2. The material properties of Inconel 152 [6] and Inconel 690 were assumed same.

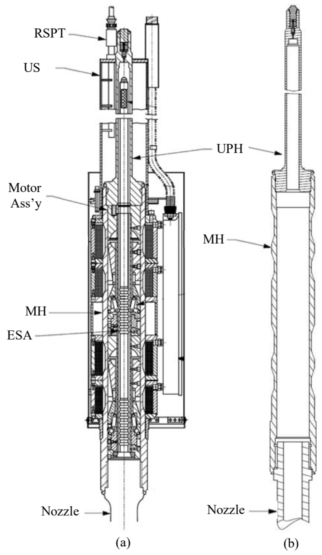

Fig. 1. Control element drive mechanism (CEDM): (a) all components, (b) pressure boundary components.

Table 1. Materials of the CEDM components.

Components	Materials
Nozzle	SB-166
MH	SA-182, F347, ASME code case N-4-13, SB-166
UPH	SA-213, TP316, SA-479, Type 316
US	304SS

2.3 Hybrid FE Model for the CEDM

Although a full solid FE model with appropriate mesh can provide accurate analysis result, computational cost is the problem. To overcome this, the goal of this study was set to develop a hybrid FE model combining solid, shell and beam elements with good accuracy. The full solid FE model of the CEDM was constructed using ANSYS program [9] for comparison and determining the range of solid and shell elements as shown in Fig. 2. The pressure boundary components and the US were considered as structural components. The 3D 8 node structural solid element, SOLID185 was used to modeling the structural components and masses of the non-structural components were distributed using the 3D 1 node structural mass element, MASS21. Mesh qualities including the aspect ratio were checked using SHPP, SUMM command and the result is shown in Fig. 2.

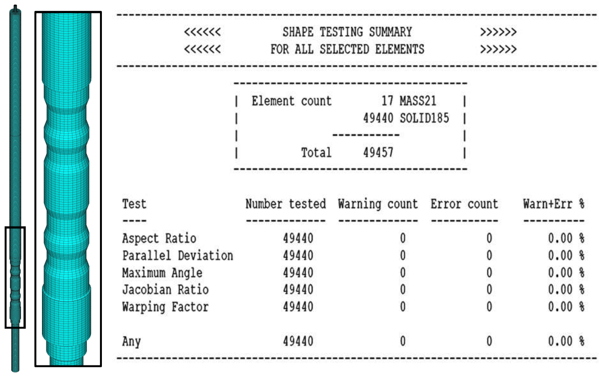

Fig. 2. Full FE model of the CEDM and mesh check result.

In order to determine the areas where the plastic stress is expected occur in dynamic condition, modal analysis was conducted with the full FE model and the areas were determined to the range of solid and shell elements for the hybrid FE model. All degrees of freedom (DOF) at bottom of the nozzle and horizontal DOFs at the elevation of the seismic support were fixed as boundary condition. Fig. 3 shows Von-Mises stress results calculated by expanding the modal analysis results.

First five modes of this analysis were considered because most of the effective masses were in these modes. Static analysis was also carried out to consider the seismic anchor motion by applying arbitrary force to the nodes at the elevation of seismic support and fixed the nodes at the bottom of the nozzle. Elements which showed stresses greater than 90% of maximum stress per components were selected as solid and shell elements and the others were changed to beam elements. Beam, solid and shell elements were connected by the multi point constraint (MPC).

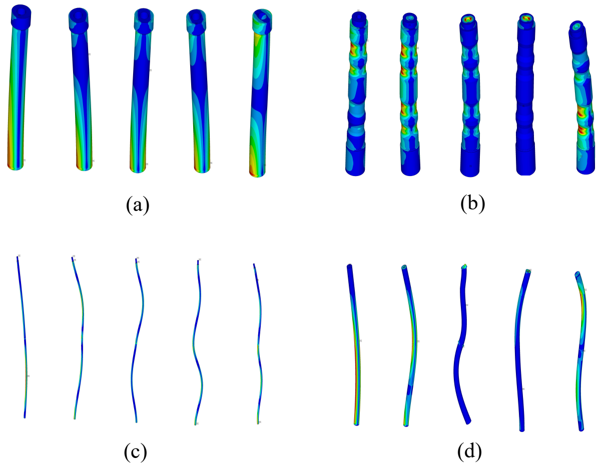

Fig. 3. Von-Mises stress results from first five modes of modal analysis results: (a) nozzle, (b) motor housing, (c) upper pressure housing, (d) upper shroud.

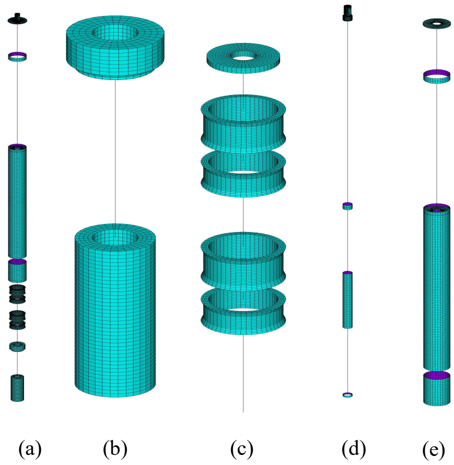

Fig. 4. Hybrid FE model of the CEDM: (a) CEDM, (b) nozzle, (c) motor housing, (d) upper pressure housing, (e) upper shroud.

Table 2. Parameters for nonlinear materials.

304 S.S	1)	σ_0	R_0	R_∞	b							
		292	0	365.6	28.78							
	2)	σ_0	C	γ								
		292	1,814.58	18.74								
316 S.S	1)	σ_0	R_0	R_∞	b							
		100	0	170	4							
	2)	σ0	C_1	γ_1	C_2	γ_2	C_3	γ_3				
		100	32,000	800	12,000	150	1,500	4				
347 S.S	2)	σ_0	C_1	γ_1	C_2	γ_2	C_3	γ_3	C_4	γ_4	C_5	γ_5
		81	60,013.1	5,671.8	21,051.2	2,931.9	52,507.3	970.7	24,309.3	230.9	1,898.1	22.2
Code Case	3)	σ_0	E_T									
N-4-13		732.9	37,921.1									
Inconel 690	1)	σ_0	R_0	R_∞	b							
		161	0	75.3	20							
	2)	σ_0	C_1	γ_1	C_2	γ_2						
		161	192,628	2,496.2	2,304.6	3.956						

Notes:
1) Voce law nonlinear isotropic hardening model [9]
2) Chaboche nonlinear kinematic hardening model [9]
3) Bilinear kinematic hardening model
4) Units for σ_0, R_0, R_∞, C are MPa

Fig. 4 illustrates the hybrid FE model developed in this study. BEAM188, SOLID185, SHELL181, and MASS21 elements were used for the hybrid FE model. Fig. 5 compares the mode frequencies of the full FE model, the hybrid model and the existing beam model. Note that the results of the beam model are most accurate as the model had been developed with experimental data. It was found that the mode frequencies of the hybrid model matched well with those of the beam model and solid model below 100 Hz and some deviations were shown over 100 Hz.

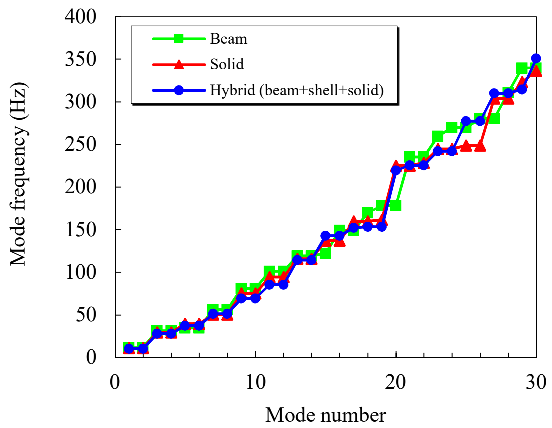

Fig. 5. Comparison of mode frequencies.

2.4 Mesh Sensitivity and Convergence Study

Ideally, the result of the analysis is more accurate as the number of element increases. However, as the number increases, computational cost also increases. Therefore, in this study, mesh sensitivity and convergence study was carried out to investigate the effect of mesh density and shape. Number of elements in radial (thickness) direction, circumferential direction and aspect ratio calculated by dividing element length in radial direction by element length in longitudinal direction of critical area of the nozzle were selected for parameters of the study. Static analysis was performed with fixed boundary condition at the bottom of the nozzle and bending force applied to the top of the nozzle. Static analysis was performed with fixed boundary condition at the bottom of the nozzle and bending force applied to the top of the nozzle. Fig. 6 shows the maximum Von-Mises stress, total equivalent strain and plastic equivalent strain result normalized by minimum value with various parameters. It was found that elements in the thickness direction shall be minimum 10 and those in the circumferential direction shall be over 96 and effect of aspect ratio, ratio of the radial length of element to the longitudinal length (e.g. large aspect ratio means more elements in the longitudinal direction), negligible but value of near 1.5 is recommended. Fig.7 illustrates the equivalent plastic strain distributions with two mesh conditions.

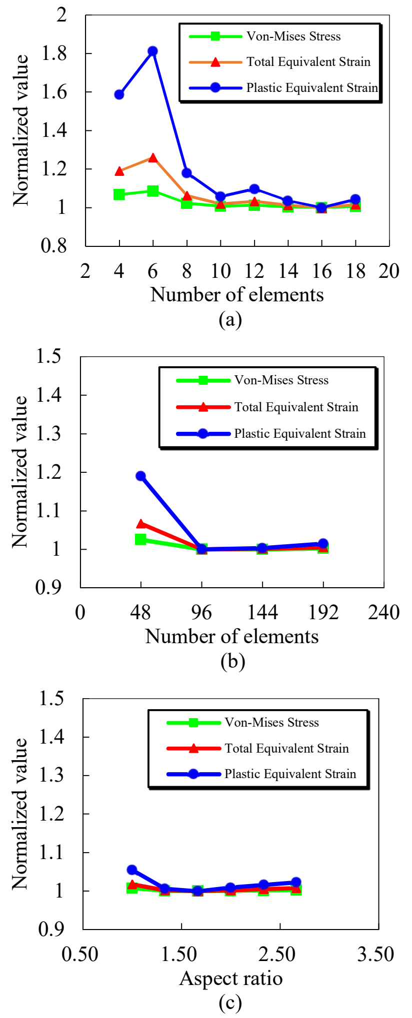

Fig. 6. Mesh sensitivity results with varying number of elements in (a) the thickness (radial) direction and (b) the rotational (circumferential) direction and (c) the aspect ratio.

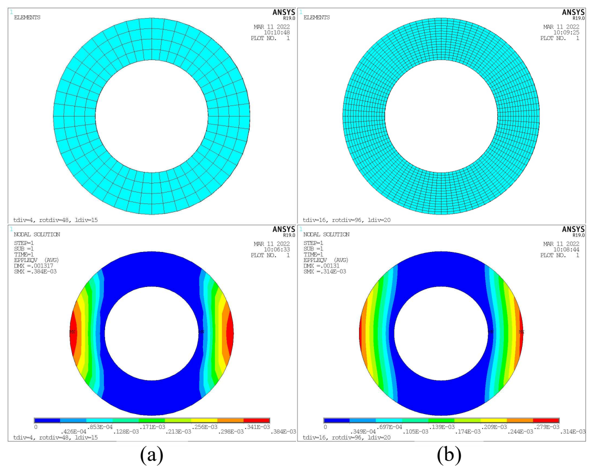

Fig. 7. Equivalent plastic strain results: number of elements in the thickness, the circumferential direction, aspect ratio; (a) (4, 48, 1.3), (b) (16, 96, 1.6)

3. Conclusions

The FE model of the CEDM for elastic-plastic analysis was developed with simplification process. Strain hardening models were applied using the data collected from literatures. Modal analysis was carried out with the FE model and the comparison of the mode frequencies showed a good conformity between the developed model and the existing model. Parametric mesh sensitivity and convergence study was carried out and required mesh density and aspect ratio were determined by static analysis.

The FE model developed in this study is expected to be used for further study to investigate the effect of material parameters, hardening models, dynamic condition, etc. and to access the structural integrity of the CEDM in BDBE condition.

REFERENCES

[1] 10 CFR 50, Appendix A, General Design Criteria for Nuclear Power Plants.
[2] ASME Boiler and Pressure Vessels Code Committee, ASME B&PV Code, Section III: Rules for Construction of Nuclear Facility Components, 2021.
[3] ASME Boiler and Pressure Vessels Code Committee, ASME B&PV Code, Code Case N-4-13.
[4] Zehsaz, M., tahami, F. F., & Akhani, H. Experimental determination of material parameters using stabilized cycle tests to predict thermal ratchetting.
[5] Schlitzer, T., Bauerbach, K., Beier, H. T., Fischaleck, M., Langschwager, K., Oechsner, M., ... & Willuweit, A. (2015). Experimental characterization and numerical assessment of fatigue crack growth under thermo-mechanical conditions, 46(2), 165-177.
[6] Muránsky, O., Smith, M. C., Bendeich, P. J., & Edwards, L. (2011). Validated numerical analysis of residual stresses in Safety Relief Valve (SRV) nozzle mock-ups. Computational Materials Science, 50(7), 2203-2215.
[7] Kim, J. Y., Lee, J. M., Park, J. G., Kim, J. S., Cho, M. K., Ahn, S. W., ... & Nam, I. K. (2022). 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(1), 343-356.
[8] ASM International, Atlas of Stress-strain Curves, 2002.
[9] ANSYS Inc., ANSYS User's Guide Ver. 19.0.