Transactions of the Korean Nuclear Society Autumn Meeting
Yeosu, Korea, October 25-26, 2018

A Study on Structural Performance of Wall-Slab Connection after Fire

Su-Hyeon Lee [a], Byong-Jeong Choi [b*], Cheol-Kyu Kang [c], Sung-Ho You [d] , Ju-Ho Kwon [e]
[a]Doctor's Course, Department of Architectural Engineering, Kyonggi Univ., 154-42, Gwanggyosan-Ro, Yeongtong-Gu, Suwon-Si, Korea, 16227
[b]Professor, Department of Architectural Engineering, Kyonggi Univ., 154-42, Gwanggyosan-Ro, Yeongtong-Gu, Suwon-Si, Korea, 16227*
[c]Assistant Professor, Department of Architectural Engineering, Kyonggi Univ., 154-42, Gwanggyosan-Ro, Yeongtong-Gu, Suwon-Si, Korea, 16227
[d]Group Supervisor, Department of Civil Engineering, Korea Electric Power Corporation Engineering and Construction(KEPCO E&C), 269, Hyeoksin-Ro, Gimcheon-Si, Gyeongsangbuk-Do, Korea, 39660
[e]Group Leader, Department of Civil Engineering, Korea Electric Power Corporation Engineering and Construction(KEPCO E&C), 269, Hyeoksin-Ro, Gimcheon-Si, Gyeongsangbuk-Do, Korea, 39660

[*]Corresponding author: bjchoi@kyonggi.ac.kr

1. Introduction

The 'Fukushima Nuclear Accident' that occurred in March 2011 increased the need to maintain structural integrity of important structures such as Nuclear Power Plants(NPPs) [1]. Even now, data on the structural behavior of large and important structures such as NPPs in the 'during fire' or 'after fire' have been lacking. It is essential to obtain fire resistance characteristics and to predict structural behavior through the analysis and theory because it is limited to experiment equipment and specifications to perform the actual fire and structural test on the large structures such as structures applied to the NPPs. Therefore, in this study, we conducted finite element analysis(FEA) and actual size of the 'Wall-Slab Connection(WSC)', which is a structure used in the 'Electrical Penetration Room(EPR)' of the NPPs in cooperation with KEPCO E&C Co., Ltd.

For the analysis, FEA program ANSYS 18.0 was used. The experiment was carried out by fire test according to ASTM E119 standard fire curve and residual structural strength test using Universal Testing Machine(UTM). For the thermal analysis can be used to temperature contour and thermal stress at each location, and structural analysis can be used to confirm the displacement and crack behavior.

2. Analysis

The analysis of the WSC was carried out using the general-purpose FEA software ANSYS(R18.0), which is capable of handling thermal gradients and highly non-linear structural problems. Temperature-dependent thermal and mechanical properties of concrete or reinforcing bar were incorporated in the analysis(Ref. Eurocode EN 1992-1-2, EN 1993-1-2) [2][3].

2.1 Thermal Model

In order to confirm the temperature contour according to the heating temperature in WSC, ASTM E119 standard fire curve was entered ANSYS(using steady state thermal)as shown in Fig.1.

As shown in Fig. 2, the cross section of the WSC was cut and temperature contour of each major time(30, 60, 120, 180min) of ASTM E119 was confirmed. It is possible to confirm the temperature contour for each location and select the thermocouple(T/C) location for the during the fire test.

As shown in Fig .3, the results of thermal-structural stress analysis according to temperature contour is shown. In this way, it is possible to confirm the stress of concrete by heating.

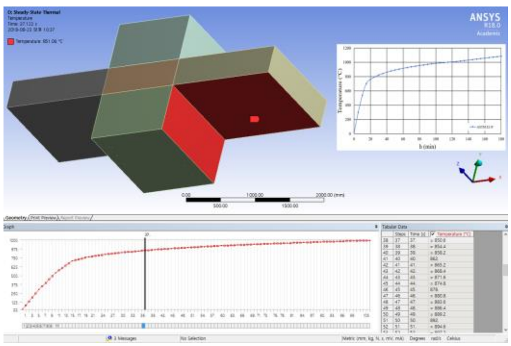

Fig. 1. ASTM E119 Standard fire curve entered into the WSC

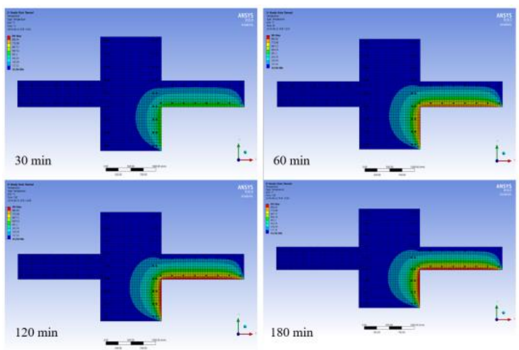

Fig. 2. Temperature contour of each major time(30, 60, 120, 180min) of WSC

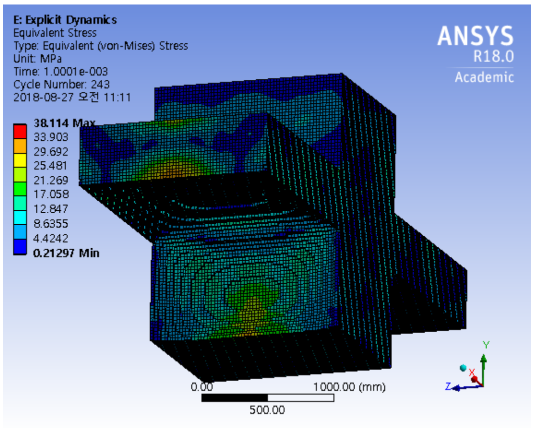

Fig. 3. Result of thermal-structural stress analysis.

2.2 Structural Model

Actuator, boundary, and WSC specimens for FEA are shown in Fig. 4.

By using the 'Explicit Dynamics' function of ANSYS, the displacement is controlled by inputting the velocity value to the actuator.

Fig. 5, Table I shows the results of the displacement analysis according to the structural analysis(non heating). When the initial crack occurred, the displacement was about 11 mm and the displacement at the end of the analysis was about 20 ~ 25 mm. Fig. 6 shows the results of WSC stress contour in the order of before stress, initial stress, crack initiation, and end of analysis. As a result of the analysis, cracks occurred at the connection part.

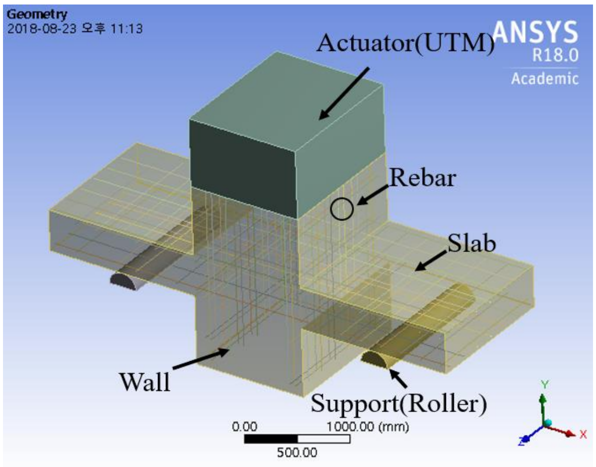

Fig. 4. Modelling of WSC

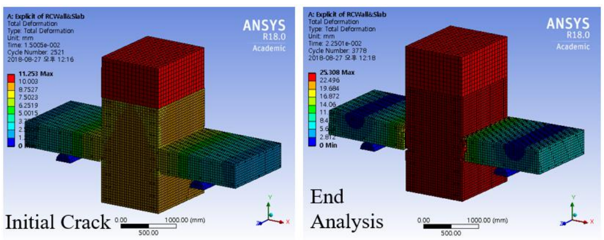

Fig. 5. Result of analysis (displacement)

Table I: Result of displacement

No.	State	Displacement (mm)
1	Initial crack	11.25
2	End analysis	20 ~ 25.3

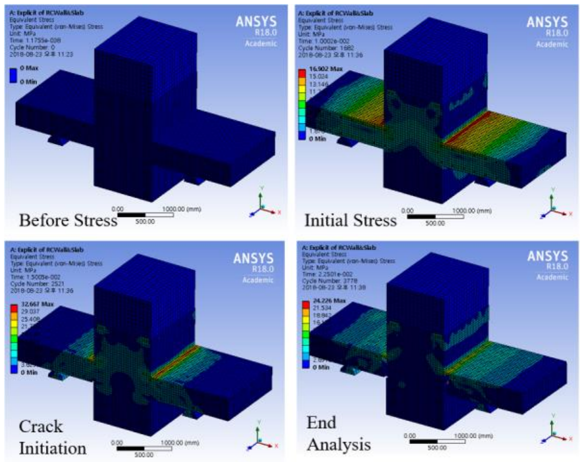

Fig. 6. Result of analysis (equivalent stress)

3. Experiment

Experiments were carried out with fire test and residual structural strength test after fire.

The fire test was carried out by one side heating using furnace, and the structural test was carried out by the residual structural strength test using UTM after fire.

3.1 Fire Test

The fire test was performed on one side heating according to ASTM E119 and set as shown in Fig. 7. The installation position of the T/C is shown in Fig.8.

The results of the fire test 1, 2, 3 hours are shown in Fig. 9 ~ 11. As shown in Fig. 8 ~ 11, the 'Spalling Effect' was observed, and temperature was rapidly increased at the portion where the reinforcing steel bar due to the spalling. Figure 12 shows the similarity of material failure due to spalling effect and thermal-structural analysis. It can be used as data for derivation of parameter in study of spalling effect according to the moisture content.

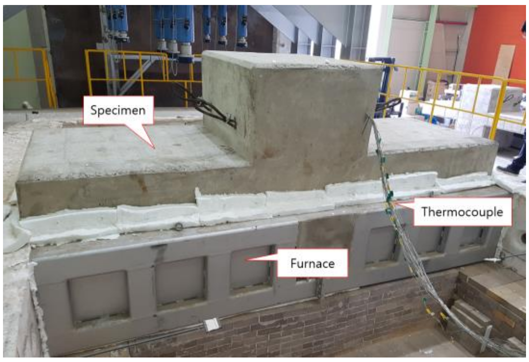

Fig. 7. Fire experiment setting

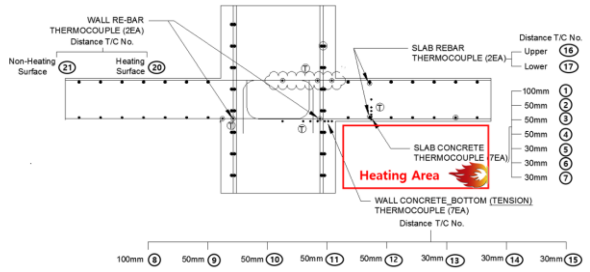

Fig. 8. Thermocouple setting when heating on one side face

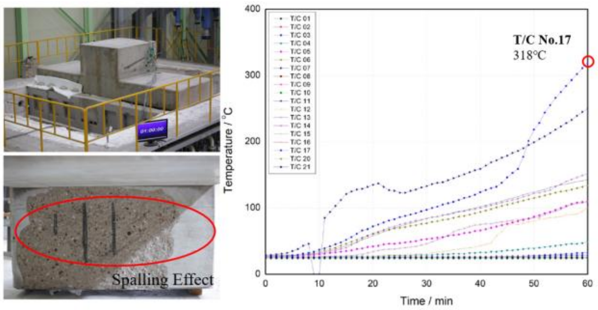

Fig. 9. Test results of 1 hour heating for one side heating

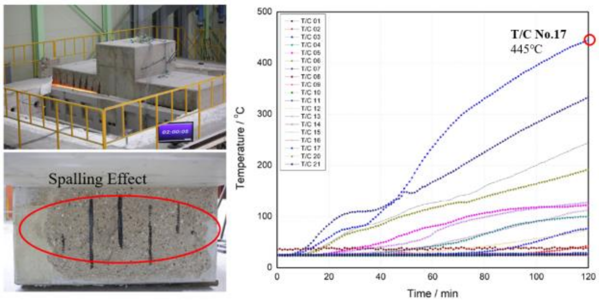

Fig. 10. Test results of 2 hours heating for one side heating

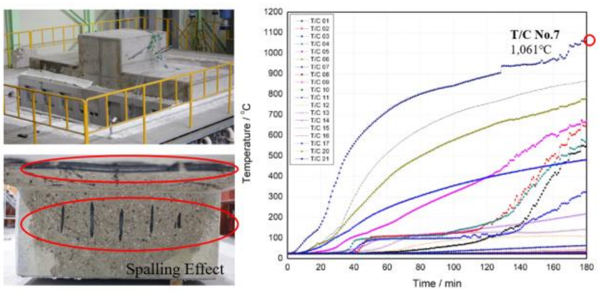

Fig. 11. Test results of 3 hours heating for one side heating

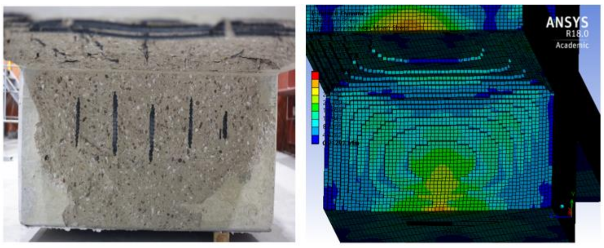

Fig. 12. Test results of 3 hours heating for one side heating

3.2 Structural Test

Structural experiments were performed using UTM and set as shown in Fig.12. The results of the structural test after fire are shown in Fig.13 after fire. The results of the 1 and 2 hour structural tests showed no significant difference from the non - heating specimens. However, as a result of 3 hours experiment, it was confirmed that the strength was decreased considerably. Table Ⅱ shows the analysis and the comparison of the experimental results of the non-heated specimen. In the non heating structure test, the displacement was 10.22mm when crack occurred, and the displacement was 11.25mm when crack occurred in analysis. At the end of the experiment, the experimental values were also derived within the range of the displacement and the analytical value.

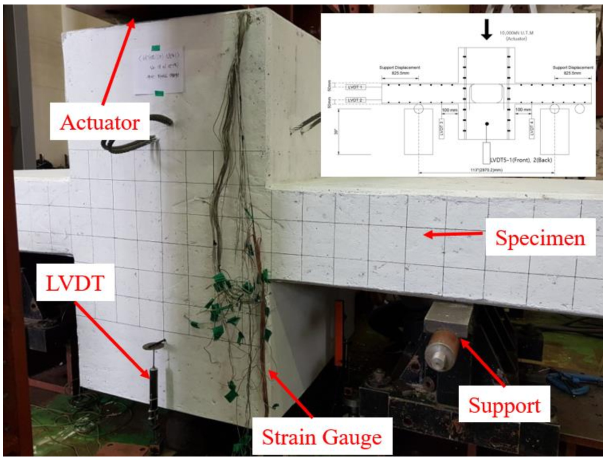

Fig. 12. Structural experiment setting

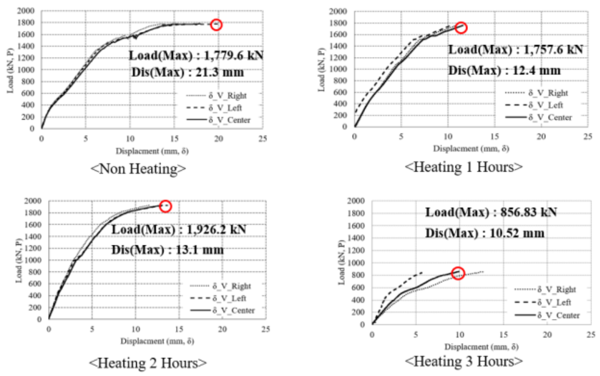

Fig. 13. Load-displacement relationship for structural test

Table Ⅱ: Comparison of analysis and structural test of non-heating specimen

Type	State	Displacement (mm)
Analysis	Initial crack	11.25
	End analysis	20 ~ 25.3
Test (non-heating)	Initial crack	10.22
	End test	21.3

4. Conclusion

In this study, the thermal-structural analysis was performed using ANSYS, a finite element analysis program, and the load-displacement relationship was derived through structural tests. In this study, parameter data for thermal-structural analysis were derived, analysis and experimental data are compared and it will be used as data for determining the residual strength for structural level with heating time.

REFERENCES

[1] Lee, S.H., (2018). Fire Resistance Performance of Steels considering fire, Graduate School of Kyonggi University, 2018.
[2] Eurocode 2(1993) Design of Concrete Structures, Part 1-2 : General rules-Structural fire design, CEN
[3] Eurocode 3(1993) Design of Steel Structures, Part 1-2 : General rules-Structural fire design, CEN

ACKNOWLEDGEMENTS

This work was supported by the Korea Institute of Energy Technology Evaluation and Planning(KETEP) and the Ministry of Trade, Industry & Energy(MOTIE) of the Republic of Korea(No.20161510400110) and implemented by scholarships for students of the Graduate School of Kyonggi University in 2018.