Development of 2D Finite Element Module to Evaluate Cladding Mechanical Behavior During Off-normal Condition

Hyo Chan Kim [a*], Yong Sik Yang [a], Jae Yong Kim [a], Young Doo Kwon [b], Yang Hyun Koo [a]
[a]*LWR Fuel Technology Division, Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseong-gu, Daejeon, 305-353, Korea*
[b]*Mechanical Engineering, Kyungpook National University, 80 Daehak-ro, Buk-gu, Daegu, 702-701, Korea*
**Corresponding author:hyochankim@kaeri.re.kr*

1. Introduction

Because the zirconium alloy cladding is the first and important barrier to prevent a fission product release, its integrity is the most important concern. In the view of a mechanical integrity, stress and strain are key factors which affect to cladding's performance during normal and or off-normal operation which induces force interaction between pellet and cladding. In order to evaluate stress and strain of cladding, one dimensional (1D) approach with various models have been proposed. [1, 2]

In case of steady state and mild transient operation period, 1D model, which considers only radial direction of cladding, is able to calculate stress and strain of cladding efficiently. However, in case of fast transient such as an Anticipated Operational Occurrence (AOO) a Reactivity Initiated Accident (RIA), it is difficult for 1D model to simulate cladding's stress and strain accurately due to its modeling limitation. Moreover, for Loss of Coolant Accident (LOCA), cladding's large deformation, so called 'ballooning', has been observed. However, this large deformation cannot be understood by 1D modeling. Consequently, Finite Element (FE) modeling which can simulate a higher order deformation of structure is indispensable requirement to understand mechanical behavior of cladding during AOO, LOCA and RIA

In this study, axisymmetric 2D FE module has been developed. The FE module employs effective-stress-function algorithm which can be stable and accurate computations of stresses in finite element thermo-elastic-plastic and creep analysis of metals. A brief description and verification results of newly developed 2D FE module are presented. To verify the developed module, commercial FE code ADINA was employed. [3]

2. Needs of FE Module

There are three mechanical forces which affect to cladding's behavior during reactor operation. Coolant pressure during whole operation period is the main force of cladding and this force result in a compressive stress to cladding. Rod internal pressure is one of the main forces to cladding, especially, higher burnup or LOCA period. Interfacial contact force generated by pellet cladding mechanical interaction can be critical severe during fast transient such as a RIA or AOO.

At steady state condition, only creep is an important permanent deformation of cladding because stresses which are loaded to cladding do not exceed yield stress of cladding. However, when fast transient or LOCA condition occurs, the applied stresses to cladding can exceed yield stress of cladding and mechanical integrity of cladding can be threatened.

Previous studies have showed that the FE analysis should be used to simulate cladding's mechanical behavior associated with experiments for off-normal operation condition. Especially, during LOCA, ballooning rupture is the key failure mode of cladding which is caused by high rod internal pressure as shown in figure 1. [4]

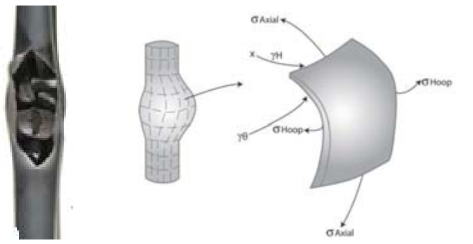


Fig. 1 Ballooning rupture and its modeling

To simulate a ballooning behavior of cladding, 2D or 3D model should be introduced. Due to the progress of computing power, higher order and large scale FE analysis are widely adopted to understand stress-strain analysis of cladding during off-normal operation condition. For this purpose, we have developed higher order FE module to simulate the cladding behavior during off-normal condition.

3. Overview of FE module

The newly developed 2D FE module employed Effective-Stress-Function (ESF) algorithm due to its efficiency and convergence. The nonlinear analysis of thermo-elastic-plastic and creep conditions has attracted much attention in research and development. It is because a stable, accurate and computationally efficient solution can be difficult to achieve with rapidly varying material conditions. The ESF algorithm solves the governing equations of the inelastic constitutive behavior by calculating the zero of the appropriate effective-stress-function: a functional relationship which involves as unknown only the effective stress. The basic incremental equations for thermo-elasto-plasticity and creep are presented. The constitutive equations can be written in the form. [5]

$$ {}^{t+\Delta t}S = \frac{{}^{t+\Delta t}E}{1+{}^{t+\Delta t}\nu}\left({}^{t+\Delta t}e' - {}^{t+\Delta t}e^{P} - {}^{t+\Delta t}e^{C}\right) \quad (1) $$

$$ {}^{t+\Delta t}\sigma_m = \frac{{}^{t+\Delta t}E}{1-2\,{}^{t+\Delta t}\nu}\left({}^{t+\Delta t}e_m - {}^{t+\Delta t}e^{th}\right) \quad (2) $$

Where for time t + Δ t

${}^{t+\Delta t}S$ = deviatoric stress tensor

${}^{t+\Delta t}e'$ = deviatoric strain tensor

${}^{t+\Delta t}e^{P}$ = plastic strain tensor

${}^{t+\Delta t}e^{C}$ = creep strain tensor

${}^{t+\Delta t}\sigma_m$ = mean stress

${}^{t+\Delta t}e_m$ = mean strain

${}^{t+\Delta t}E$, ${}^{t+\Delta t}\nu$ = Young's modulus and Poisson's ratio

${}^{t+\Delta t}e^{th}$ = thermal strain

4. Evaluation

To verify the developed FE module, code-to-code benchmark was performed by using of commercial FE code ADINA under same conditions.

The evaluation does not consider thermal load and creep yet. Four quadrilateral elements which include 8 nodes were generated as shown in Figure 2. Plasticity behavior is defined as bilinear curve presented by elastic modulus and plastic modulus. Figure 2 also shows load, boundary conditions and time step for FE simulation.

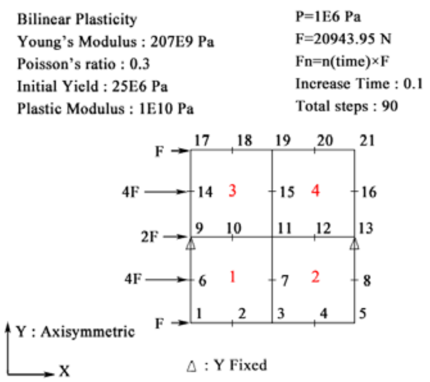
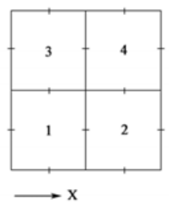


Fig. 2 Evaluation conditions

For the evaluation, ADINA which has been widely used as reliable and efficient Finite Element procedure was employed. With identical conditions of FE module, initial yield stress and strain hardening were inserted to ADINA. As a result, Figure 3 shows comparison of Von Mises stresses on each element at 90 steps. Consequently, this agreement shows that the FE module is acceptable in terms of mechanical nonlinear behavior. To improve model accuracy, validation of the model should be carried out based on experimental data.



Von Mises (MPa)	Fortran	ADINA
1	25.39	25.39
2	25.00	25.00
3	25.39	25.39
4	25.00	25.00

Fig. 3 Comparison of von mises stress on each element

5. Discussion

Axisymmetric 2D FE module has been developed to simulate mechanical behavior of cladding during off-normal operation condition. The FE module was evaluated in terms of mechanical nonlinear behavior by comparison of commercial FE code results. Based on simulation results, the von mises stress of each element from FE module results shows a good agreement against that of ADINA. It demonstrates that algorithm of the module is acceptable to simulate plastic behavior with FE model.

For the future, the FE module should be verified with various test cases using commercial FE code. Validations of the module are also planned with various experimental data. Furthermore, development of 3D FE module which can be applied to fast transient such a RIA and AOO is progressing.

REFERENCES

[1] J. S. Cheon, B. H. Lee, Y. H. Koo, J. Y. Oh, D. S. Sohn, Evaluation of a pellet-clad mechanical interaction in mixed oxide fuels during power transients by using axisymmetric finite element modeling, Nuclear Engineering and Design, Vol.231, p. 39, 2004.

[2] N. Marchal, C. Campos, C. Garnier, Finite Element Simulation of Pellet-Cladding Interaction (PCI) in nuclear fuel rods, Computational Material Science, Vol.45, p.821, 2009.

[3] ADINA R&D Inc., ADINA manuals, 2008.

[4] E. Kolstad, Overview of LOCA tests at halden status and plans, Workshop on VVER LOCA testes at Halden, Helsinki, 2006.

[5] M. Kojic, K. J. Bathe, The 'Effective-Stress-Function' Algorithm For Thermo-Elasto-Plasticity and Creep, International Journal For Numerical Methods In Engineering, Vol.24, p. 1509, 1987.