Evaluation of Nonlinear Finite Element module for the Simulation of Pellet and Cladding Mechanical Interaction

Hyo Chan Kim [a*], Yong Sik Yang [a], Dae Ho Kim [a], Je Geon Bang [a], Sun Ki Kim [a], Jae Yong Kim [a], Yang Hyun Koo [a], Young Doo Kwon [b]
[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

The core of nuclear Light Water Reactors (LWRs) consists of fuel assemblies that contain hundreds of fuel rods. And fuel rods consist of zirconium alloy tubes containing uranium dioxide pellets. Because the zirconium alloy cladding is the most critical containment barrier for fission products released into the environment, its mechanical integrity is the most important concern. In view of mechanical integrity, stress and strain are the main factors that affect the cladding performance during normal or off-normal operation, which induces force interaction between pellet and cladding as shown in Figure 1.

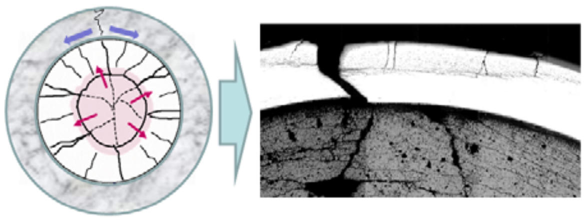


Fig. 1 Cladding failure by pellet and cladding mechanical interaction

Many researchers have developed a fuel performance code with a one-dimensional (1D) approach so as to investigate the stress and strain of cladding during operation. In the case of a steady state and mild transient operation period, a 1D model that considers only the radial direction of the cladding is able to calculate the stress and strain of the cladding efficiently. However, in the case of a fast transient such as an Anticipated Operational Occurrence (AOO), it is difficult for a 1D model to simulate the stress and strain of the cladding accurately due to its modeling limitation. Moreover, for a Loss of Coolant Accident (LOCA), the cladding's large deformation, a so-called 'ballooning', has been observed. This large deformation along the radial and axial directions cannot be understood by 1D modeling. Consequently, Finite Element (FE) modeling, which can simulate a higher degree of freedom, is an indispensable requirement to understand the mechanical behavior of cladding during off-nominal operation.

Previous studies have shown that an FE analysis should be introduced to simulate the cladding's mechanical behavior associated with experiments for off-normal operation conditions. In particular, during LOCA, a ballooning rupture is a key failure mode of cladding which is caused by high rod internal pressure. To simulate the ballooning behavior of the cladding, a 2D or 3D FE model should be introduced. [1, 2]

In this work, a two-dimensional FE module, which will be integrated into a transient fuel performance code, has been developed. To solve the mechanical equilibrium of the pellet-cladding system taking into account the geometrical and material non-linearities, the FE module employs an effective-stress-function (ESF) algorithm, which can acquire stable and accurate computations. The fundamental theory of the ESF algorithm and an overall flowchart of the FE module were also studied. ANSYS 13.0, which is widely used in the mechanic fields, was employed as a commercial FE code. Verifications of the FE module for thermal and elastic analyses were performed using the results of ANSYS 13.0. Consequently, this demonstrates that the developed FE module is acceptable and reasonable.

2. Implementation of Nonlinear FE Module

The basic incremental equations for thermo-elasto-plasticity-creep formulate the effective-stress-function algorithm for von Mises elasto-plasticity with isotropic hardening. Including thermo-elastic-plastic and creep deformations, the constitutive equations can be written as follows.

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

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

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

Taking the scalar product of the formulas, we obtain

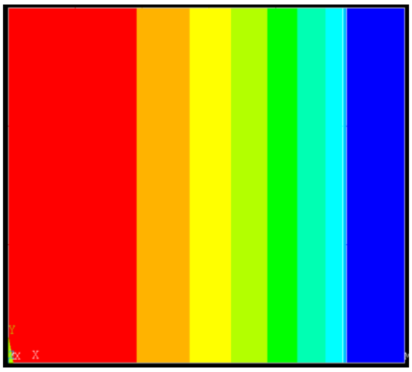
$$f({}^{t+\Delta t}\overline{\sigma}) = a^2({}^{t+\Delta t}\overline{\sigma})^2 + b^{\tau}\gamma - c^{2\,\tau}\gamma^2 - d^2 \qquad (3)$$

The coefficients b, c and d are constants that depend only on the known values, independent of ${}^{t+\Delta t}\overline{\sigma}$, whereas the value of the coefficient 'a' varies with ${}^{t+\Delta t}\overline{\sigma}$. [3]

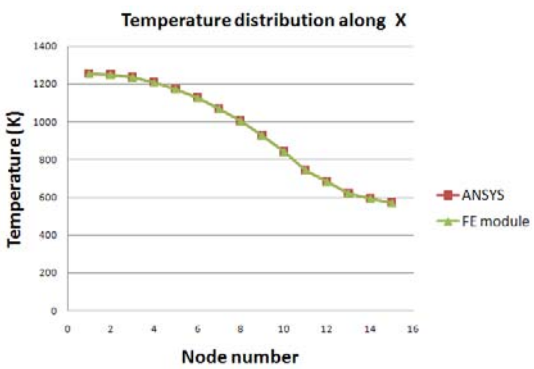
3. Evaluation

To verify the developed FE module, a code-to-code benchmark was performed using commercial FE code ANSYS 13.0, which has been widely used as a reliable and efficient Finite Element code. [4]

Twenty one axisymmetric quadrilateral elements, which include 8 nodes, were generated. The dimensions of the geometry are as follows: height, 4.9 mm; pellet radius, 5.0 mm; gap thickness, 0.06 mm; and cladding thickness 0.87 mm.



(a) Temperature distribution



(b) Comparison of temperature at node

Fig. 2 Comparison of the thermal analysis results

The assumptions are as follows: all materials were isotropic, the gap was considered as a solid material, and thermal contact effect was neglected. To calculate the temperature distribution, the boundary conditions are as follows: the body heat generation of the pellet is 327.62 * 10^6 W/m^3. Convection heat loss was only applied to the cladding outer surface. Convection coefficient and reference temperature are 1 * 10^5 W/m^2 K and 562.7 K, respectively. Using the material properties and boundary conditions, the temperature distribution can be calculated. Node temperatures at the bottom plane were compared with the results derived from ANSYS. The temperature of each node from the FE module shows good agreement against that of ANSYS, as shown in Figure 2(b). This demonstrates that the thermal module of the developed code is acceptable.

4. Conclusion

Nonlinear FE module has been developed to simulate the mechanical behavior of the cladding and pellet for transient conditions. The newly developed 2D FE module should include thermo-elastic-plastic-creep analyses as the pellet and cladding during off-normal operation are loaded by high hydrostatic pressure at high temperature. To resolve the complex finite element problem of thermo-elastic-plastic and creep, the Effective-Stress-Function (ESF) algorithm was adopted. To verify the developed FE module, a code-to-code benchmark was performed using commercial FE code ANSYS 13.0, which has been widely used as a reliable and efficient Finite Element code. Based on comparison results from thermal and mechanical analyses, the algorithm of the FE module is acceptable and reasonable.

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] 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.

[4] ANSYS R&D Inc., ANSYS manuals, 2010.

[5] H. C. Kim, Y. S. Yang, J. Y. Kim, Y. D. Kwon, Y. H. Koo, Development of 2D Finite Element Module to Evaluate Cladding Mechanical Behavior during Off-normal Condition, Transactions of the Korean Nuclear Society Autumn Meeting, Gyeongju, Korea, p.255, 2011.