Development of NEPTUNUS for simulation of fuel bundle behavior during LOCA

Jinsu Kim[a], Taek Jin Jang[a], Choong Myoung Lee[a], Hyochan Kim[b], Sung-Uk Lee[b], JaeYong Kim[b], Jeong Whan Yoon[a,*],
[a]*Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology, 291 Daehak-ro, Yuseong-gu, Daejeon, 34141, Republic of Korea*
[b]*Advanced 3D Printing Technology Development Division, Korea Atomic Energy Research Institute, 111 Daedeok-daero 989 beon-gil, Yuseong-gu, Daejeon, 34057, Republic of Korea*
**Corresponding author: j.yoon@kaist.ac.kr*

1. Introduction

During Loss of Coolant Accident (LOCA), a cladding undergoes ballooning and burst failure due to increased temperature and inner pressure. This thermo-mechanical behavior is one of major concerns in nuclear fuel safety criteria, and hence various numerical studies were conducted to predict the behavior. However, most of the numerical studies were mainly focused on a single rod's behavior which is different with the actual fuel assembly where fuel bundle exists rather than the single rod. Hence more detailed analysis which considers the fuel bundle is required to predict the cladding's behavior accurately.

When the LOCA occurs in the fuel bundle, the cladding's behavior is different from the single rod behavior due to the interactions between the bundles. Heat transfer from the neighboring rods to the cladding occurs through the radiation heat transfer, which accelerates the cladding temperature rise. When the cladding's ballooning reaches some extent, contact between the cladding and the neighboring rods occurs and it affects its thermo-mechanical behavior. For example, bursting failure can be delayed and heat transfer through contacted surface takes place. Moreover, this contact interaction affects the coolant flow in the channel, which ultimately leads to flow blockage during the accident. Therefore, it is necessary to take into account fuel rod interactions in the accident analysis to more accurately predict the fuel rod in the actual fuel assembly.

In this study, three dimensional(3D) fuel rod analysis code, which is called NEPTUNUS (**N**uclear fu**E**l bundle **P**erformance code wi**T**h FE sim**U**latio**N** methodology and analytical form**U**la**S**), has been developed to describe the fuel rod bundle's behavior during the LOCA. The code is based on coupled thermo-mechanical finite element analysis. It employs 3D shell element to model the thin-walled cladding tube to reduce computational cost incurred by modeling the entire fuel rods with solid elements. The code incorporates interaction modules to model the radiation, mechanical and thermal contact, as well as gap conductance and high temperature creep to predict thermo-mechanical behavior of the cladding.

2. Methods and Results

2.1. Structure of the code

The NEPTUNUS performs thermo-mechanical coupled analysis based on transient heat transfer and structural dynamic analysis. It is composed by coupling two analysis modules which are TRITON (**T**he**R**mal analysis module **I**ncluding **T**hermal c**ON**tact) for the transient heat transfer analysis, and DAEC (**D**ynamic **A**nalysis **E**xplicit **C**ode) which was developed in reference [1] for the structural dynamic analysis. Figure 1 shows a relationship between the two modules in the NEPTUNUS and constituents that consist of each module.

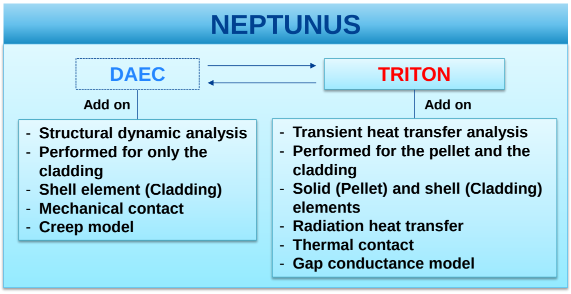

Fig. 1. Structure of NEPTUNUS and each module.

The transient heat transfer analysis by the TRITON is performed for both of the pellet and the cladding since heat transfer from the pellet to the cladding has a significant impact on the accident. During the heat transfer analysis by the TRITON, temperature field is obtained including radiation heat transfer and thermal contact, and gap heat transfer is also carried out by using gap conductance model.

The structural analysis by the DAEC is performed only about the cladding since other parts except for the cladding can be assumed to be the rigid bodies. When the DAEC calculates mechanical behavior of the cladding, it calculates displacement, strain and stress caused by external loads and temperature change as well as the mechanical contact and the creep behavior.

Note that structural analysis methodology used in this study requires smaller time step size than heat transfer. Thus, an appropriate time stepping scheme is necessary for computational efficiency. A graphical illustration of the time stepping scheme in this code is shown in Fig. 2.

If the appropriate time step size Δt is determined, heat transfer analysis is firstly performed by using the time step which is 100 times larger than Δt. Then structural analysis is performed by using Δt. During each structural analysis step, temperature field is linearly interpolated from the results of previous and current steps.

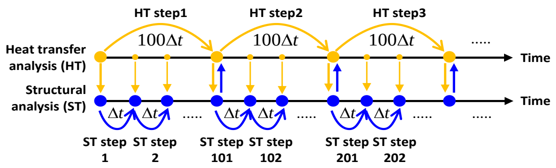

Fig. 2. Time stepping scheme.

In addition, the cladding is modeled using shell element while the pellets are modeled with standard solid element. So, shell element is also required to be formulated and implemented into the code for both of the heat transfer and structural analyses. In this study, curved shell element which is presented by Surana and Abusaleh [2] for heat transfer analysis was employed. For the structural analysis, to prevent element locking in large deformation and enhance computational efficiency, one-point quadrature shell element proposed by Cardoso et al. [3] was used.

2.2. Modeling of interactions between fuel rod bundle

To consider the interactions between the fuel rod bundle which are radiation heat transfer, mechanical and thermal contact, it is necessary to construct and implement modules for each behavior.

In radiation heat transfer, it is necessary to account for the effect of orientation between radiation source and target, and thus a view factor is introduced. The view factor is a fraction of the radiation energy leaving the source surface to that directly striking the target surface and is able to quantify the effect of geometric effect such as shapes, orientation, etc. In the fuel rod bundle analysis, the view factor for the parallel cylinders [4] is used. Note that since the radiation is a surface phenomenon, the cylinder view factor can be used for the cladding although it is a tube. After obtaining the view factor, radiosity which is total radiation heat flux leaving a surface is calculated for each surface with emissivity and temperature of the cladding and neighboring rods. Lastly, net radiation energy of each surface is obtained from the radiosity and net heat flux is imposed as a boundary condition in the heat transfer analysis.

To consider the contact between the cladding tubes, the mechanical contact module was implemented by taking a module implemented in DAEC [1]. The contact module is composed of three steps, contact pair search, penetration judgment, and application of penetration force. In the search step, the NEPTUNUS checks the contact pair where the contact is likely to occur. Next, it judges whether a slave node penetrates a master surface in the contact pair or not. If the slave node penetrates the master surface, reaction force is calculated to push both the slave node and master surface oppositely. An equation of the reaction force is as follows:

$$F_c = f_{pen}\left(\frac{p-(v_s-v_m)\Delta t}{\Delta t^2}-\frac{f_{int1}}{M_1}+\frac{f_{int2}}{M_2}\right)\frac{M_1 M_2}{M_1+M_2} \quad (1)$$

where F_c is the reaction force due to the contact, f_{pen} is correction factor, p is penetrated displacement, (v_s, M_1) and (v_m, M_2) are velocity and mass of the slave node and the master surface, respectively. f_{int1} and f_{int2} are the internal forces applied to the slave node and the master surface, respectively.

The thermal contact module is activated when the mechanical contact occurs and its calculation procedure is illustrated in Fig. 3.

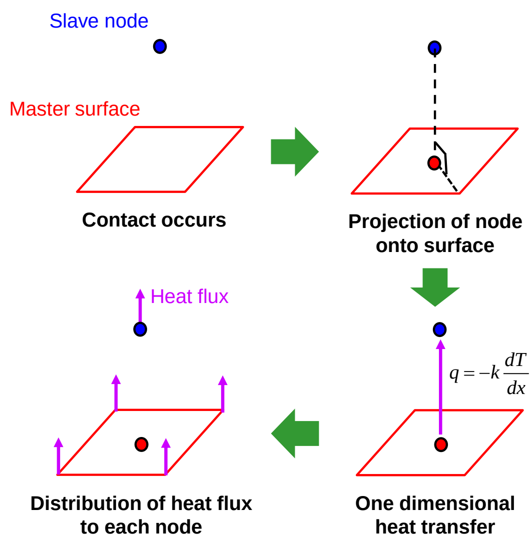

Fig. 3. Procedure of the thermal contact analysis.

The thermal contact module gets data such as contact pair information, shape function, and contact pressure from the mechanical contact module. Then, the slave node of the contact pair is projected onto the master surface. The heat flux can be calculated as one-dimensional heat transfer relationship [5]. The equation of the one-dimensional heat transfer is as follows:

$$q=-k\frac{dT}{dx} \quad (2)$$

where q is heat flux, k is thermal contact conductance, T is the temperature, and x is the distance.

The calculated heat flux is distributed to both of the slave and the master surface nodes, and then the applied heat flux is imposed as the boundary condition in the heat transfer analysis.

2.3. Gap conductance and creep model

Because heat energy generated in the pellet is transferred to the cladding through the gap between them, it is required to model the gap heat transfer by using a gap conductance model. The NEPTUNUS employs Ross and Stoute gap conductance model [6] as expressed in equation (3).

$$h_{\text{gap}} = \frac{k_{\text{gas}}}{\Delta x + g + \Delta R} \tag{3}$$

where k_{gas}, Δx, g, and ΔR are the gas conductivity of the internal gas, distance between the pellet and the cladding, temperature jump distance, and roughness of each surface, respectively.

In addition, the cladding undergoes high temperature creep behavior during the LOCA, which results in ballooning and burst. The present study employs Norton-Bailey creep model, which is expressed in equation (4), to model the cladding's creep behavior.

$$\dot{\varepsilon}^{cr} = A\exp\left(-\frac{Q}{RT}\right)\bar{\sigma}^{n} \tag{4}$$

where $\bar{\sigma}$ is an effective stress, R is universal gas constant (8.314 J/mol·K), T is temperature. Coefficients A, n, and Q which is activation energy (unit: J/mol) are taken from Rosinger [7].

3. Conclusions

In this study, a three dimensional fuel rod analysis code known as NEPTUNUS has been developed to describe thermo-mechanical behavior of fuel rod bundle and resulting flow blockage during the LOCA. It is based on coupled thermo-mechanical analysis through transient heat transfer and dynamic analyses. Also, it utilizes shell element to model the cladding to improve computational efficiency. To model interactions that occur in the fuel bundle, the radiation heat transfer module was implemented. The mechanical contact module was taken from the previous study and thermal contact module was formulated and implemented. It also incorporates gap conductance and high temperature creep model.

Acknowledgement

This work has been carried out under the Nuclear R&D Program supported by the Ministry of Science and ICT (NRF-2017M2A8A4015024).

REFERENCES

[1] KAERI/CM-2630/2018, Development of a Bulge Joint Detail Model and Analysis Modules for Dynamic Analysis of a Fuel Component, Korea Atomic Energy Research Institute, 2018.

[2] K. S. Surana and G. Abusaleh, Curved shell elements for heat conduction with p-approximation in the shell thickness direction, Computers and Structures, Vol. 34, No. 6, pp. 861-880, 1990.

[3] R. P. R Cardoso et al., A new approach to reduce membrane and transverse shear locking for one-point quadrature shell elements: linear formulation, International Journal for Numerical Methods in Engineering, Vol. 66, No. 2, pp. 214-249, 2006.

[4] F. Incropera et al., Fundamentals of heat and mass transfer, John Wiley & Sons, 2006

[5] P. Wriggers and C. Miehe, Contact constraints within coupled thermomechanical analysis-A finite element model, Computer Methods in Applied Mechanics and Engineering, Vol. 113, No. 3–4, pp. 301–319, 1994.

[6] A. Ross and R. Stoute, Heat transfer coefficient between UO_2 and Zircaloy-2, Atomic Energy of Canada Limited, 1962.

[7] H. E. Rosinger, A model to predict the failure of Zircaloy-4 fuel sheathing during postulated LOCA conditions, Journal of Nuclear Materials, Vol. 120, No. 1, pp. 41-54, 1984