Thermal Ratcheting Analysis of the High-Temperature Reactor Vessel using CalculiX

Jin Haeng Lee [a*], Jonggan Hong [a]
[a]*Korea Atomic Energy Research Institute, 111, Daedeok-daero 989 beon-gil, Yuseong-gu, Daejeon, Korea*
[*]*Corresponding author: jinhaeng@kaeri.re.kr*

1. Introduction

In the nuclear reactors operated in high-temperature conditions, the structural integrity are complexly affected by the interaction of fluids and structures. In particular, nonlinear material plastic properties, complex geometries, and boundary conditions make structural analysis difficult, and therefore excessive conservativeness may be included in the analysis results during the simplification process. In order to solve these problems, various attempts have recently been made on Fluid-Structure Interaction (FSI) analysis.

An open-source code OpenFOAM for fluid analysis, can be linked with various structural Finite Element Analysis (FEA) codes using preCICE [1]. Among them, another open-source code CalculiX [2], which provides an implicit/explicit solver CalculiX CrunchiX (CCX), is capable of nonlinear large deformation FEA. Since CalculiX is very similar in input format and basic functions to the commercial FEA code, Abaqus [3], it has good accessibility and can be usefully applied to FSI coupled analysis.

In this study, to check the validity of structural analysis including plastic deformation and the applicability of FSI coupled analysis using CalculiX, the thermal ratcheting phenomenon of the reactor vessel of the Sodium-cooled Fast Rector (SFR) is simulated by using Abaqus and CalculiX. The analysis procedures of them are reviewed, and the characteristics of CalculiX for nonlinear plastic analysis are examined by comparing the results of Abaqus.

2. Thermal Ratcheting Caused by Moving Temperature Front

Ratcheting is a progressive plastic deformation caused by unsymmetric mechanical or thermal cyclic loads. There are two types of ratcheting: material ratcheting and structural ratcheting. Material ratcheting occurs under a relatively uniform stress distribution, such as a uniaxial cyclic loading test. On the other hand, structural ratcheting occurs when the structure has a stress gradient especially caused by secondary thermal stress, which is usually divided into Bree-type typical thermal ratcheting and thermal ratcheting by moving temperature front [4-5]. Typical thermal ratcheting occurs under a combination of a constant primary stress and a cyclic thermal stress. Thermal ratcheting caused by moving temperature front occurs in areas where a temperature front having large temperature gradient periodically moves. A representative example is the reactor vessel of high-temperature reactor. During the startup and shutdown of the reactor, the primary coolant temperature and its level in the reactor vessel change, which generates the circumferential membrane stress. The reactor vessel can periodically expand along the radial direction owing to the ratchet deformation accumulated by this stress change. This kind of ratcheting does not require a primary stress, and it occurs when sufficient temperature difference and moving distance of the temperature front are simultaneously accompanied.

3. Finite Element Analysis Procedures

3.1 FE modeling

The core inlet/outlet temperatures of the SFR used in the present work were 360/510°C. The space above the sodium coolant in the reactor vessel was assumed to be filled with inert argon gas.

Figure 1 shows the geometry and thermal boundary conditions of the reactor vessel wall. The upper part of the reactor vessel at a distance of 8.2 m from the bottom of the reactor head was used for the finite element model. The outer radius and thickness of the reactor vessel were 8750 mm and 25 mm, respectively. The sodium coolant level in normal operation (Normal Service Level, NSL) was set as the reference position.

The reactor vessel wall was modeled using CAX8(T), a two-dimensional axisymmetric 8-node element. The total numbers of elements and nodes were 5000 and 17011, respectively, and 5 elements were used in the thickness direction. The displacements of the upper nodes of the model were completely fixed, and the circumferential deformation and uniform deformation in the axial direction were possible at the lower nodes. The temperature at the top was fixed at 100°C, and the convective heat transfer coefficients and bulk temperatures given to the inner surface were assigned by location as shown in Fig. 1. The outer surface of the reactor vessel was assumed to be completely insulated.

The material of the reactor vessel was 316 stainless steel, and the physical properties of which described in the ASME code considering the effect of temperature change were used. In the plastic region, perfectly plastic material model was used to increase the efficiency of analysis by applying a conservative and simple model.

The coolant temperatures were assumed to be 200°C in the initial and reloading states and 510°C in normal operation. Figure 2 shows the changes in sodium coolant level and temperature with time applied to the

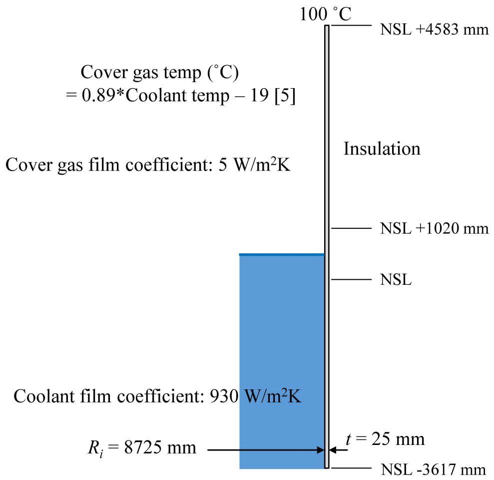

Fig. 1. The geometry and thermal boundary conditions of the reactor vessel wall of the SFR

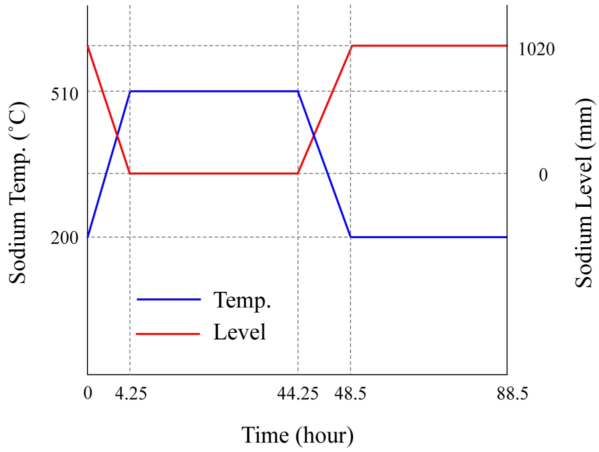

Fig. 2. The cycles of the sodium coolant temperature and its level

analysis. Although there is currently no requirement for the time taken for startup and shutdown of the reactor, the transient time was set much shorter than the expected time to artificially increase the effect of plastic deformation in the analysis.

The temperature distribution, displacement, and stress of the reactor vessel were investigated through the analysis of 10 repetitions of the four steps (startup-hold-shutdown-hold) shown in Figure 2. Accordingly, a total of 41 steps including the steady state step for the initial temperature condition were analyzed.

2.2 Coupled thermal-structural analysis

When performing thermal-structural analysis, both Abaqus and CalculiX codes can apply sequentially or fully coupled analysis.

The sequentially coupled analysis is a method that includes the temperature field calculated by the heat transfer analysis in the stress analysis. HEAT TRANSFER keyword was used for heat transfer analysis, and convective boundary conditions were assigned to the boundary elements using FILM keyword applicable to both Abaqus and CalculiX. In these two FE codes, the temperature field file can be included in the stress analysis using the predefined field. In addition, only in CalculiX, there is a keyword called UNCOUPLED TEMPERATURE-DISPLACEMENT. When this keyword is used, heat transfer analysis is first performed for each increment, and stress analysis is performed in the same increment using the obtained temperature field.

The fully coupled thermal-structural analysis is a method that simultaneously performs stress, displacement, and temperature field analysis. Both Abaqus and CalculiX codes can use COUPLED TEMPERATURE-DISPLACEMENT keyword for the fully coupled analysis. To perform the coupled thermal-structural analysis in Abaqus, a coupled temperature-displacement element should be used. Hence, for the axisymmetric 8-node element in this example, a CAX8T element must be used instead of CAX8. On the other hand, CalculiX can use the CAX8 element as it is for the fully coupled analysis.

In this study, the characteristics of Abaqus and CalculiX were compared through sequentially/fully coupled analysis by using UNCOUPLED TEMPERATURE- DISPLACEMENT keyword for CalculiX and COUPLED TEMPERATURE-DISPLACEMENT keyword for Abaqus and CalculiX.

The amount of plastic deformation caused by thermal ratcheting of the SFR reactor vessel is determined by the range of temperature change, the travel length of the temperature front, and the number of repetitions. User subroutines were used in the FILM keywords of Abaqus and CalculiX to simultaneously consider changes in sodium coolant level and temperatures of coolant/cover gas during the transient time. Since the coolant level and temperatures of coolant/cover gas of a specific time are known from Fig. 2, it is possible to assign a convective boundary condition for each node on the boundary element at the specific time and step. Here, the medium around the boundary element changes depending on the coolant level, so user subroutines were written to take it into account. The user subroutine code of CalculiX is almost identical to that of Abaqus, but there exist some differences. In particular, when the user subroutine of CalculiX is modified, compiling for CalculiX CCX is required.

3. Comparison of the Abaqus and CalculiX analysis results

The thermal ratcheting phenomenon caused by the temperature front movement was analyzed using Abaqus and CalculiX, and the validity of the CalculiX code was verified by comparing the results.

3.1 Comparison of the calculation time

In parallel analysis using 64 CPUs, it took 37 and 497 minutes, respectively, for the fully coupled thermal-structural analysis using the COUPLED TEMPERATURE-DISPLACEMENT keyword of Abaqus and CalculiX on the Windows system. The analysis time of CalculiX is about 13.4 times longer than that of Abaqus. In the case of the sequentially coupled analysis applying the UNCOUPLED TEMPERATURE-DISPLACEMENT keyword of CalculiX, it took 551 minutes, which is 14.9 times longer than that of fully coupled analysis time of Abaqus. The main reason for the longer analysis time of CalculiX compared to Abaqus is that a large number of iterations are required within one increment and a relatively small time increment lasts.

3.2 Comparison of the temperature, displacement, and stress fields

Figure 3 shows the temperature distribution and deformed shape in step 1 ~ 5, and Table I summarizes the maximum temperature calculated in each step. In Fig. 3, the scale factor of 30 was applied collectively. Since the deformed shapes of sequentially and fully coupled thermal-structural analyses using CalculiX are almost identical, Figure 3 compares only the results for the fully coupled analysis. The temperature distribution and deformation patterns for the following cycles are repeated almost identically to the results for the first cycle. From the change in the temperature distribution for each step, it can be seen that the boundary conditions for thermal analysis were properly applied.

Figure 3(a) shows the temperature distribution before starting the reactor which is the same as the reloading state. The temperature at the bottom of the reactor head is fixed at 100ºC, and the maximum coolant temperature is 200ºC, so a temperature distribution is formed between these two values.

In step 2, the liquid level decreases as the coolant temperature rises to 510ºC. The maximum temperature of the vessel rises to a level similar to the coolant temperature because of the high heat transfer coefficient of coolant. The maximum temperatures of the reactor vessel for the three analyses are 507.5, 507.6, and 507.6ºC, as shown in Table I, and there is little difference. In Fig. 3(b), as the temperature difference between the upper and lower parts of the reactor vessel increases, it can be seen that the radial displacement of the vessel located below the coolant level is relatively large because of the difference in thermal expansion.

In step 3, while the fluid (coolant and cover gas) temperatures and coolant level of step 2 are maintained, the maximum wall temperature rises to 510ºC as listed in Table I, which is the same as the coolant temperature. Although the maximum temperature of the vessel does not change significantly from the previous step, it can be seen from that the increase in radial displacement extends to the upper part of the vessel as heat transfer steadily continues.

Step 4 is a step back to the boundary conditions used in step 1 as the fluid temperatures decrease and the coolant level rises. The lower part of the vessel in contact with the coolant has a large amount of displacement recovery caused by the temperature drop, whereas the cover gas region having a low heat transfer coefficient undergoes relatively less heat shrinkage.

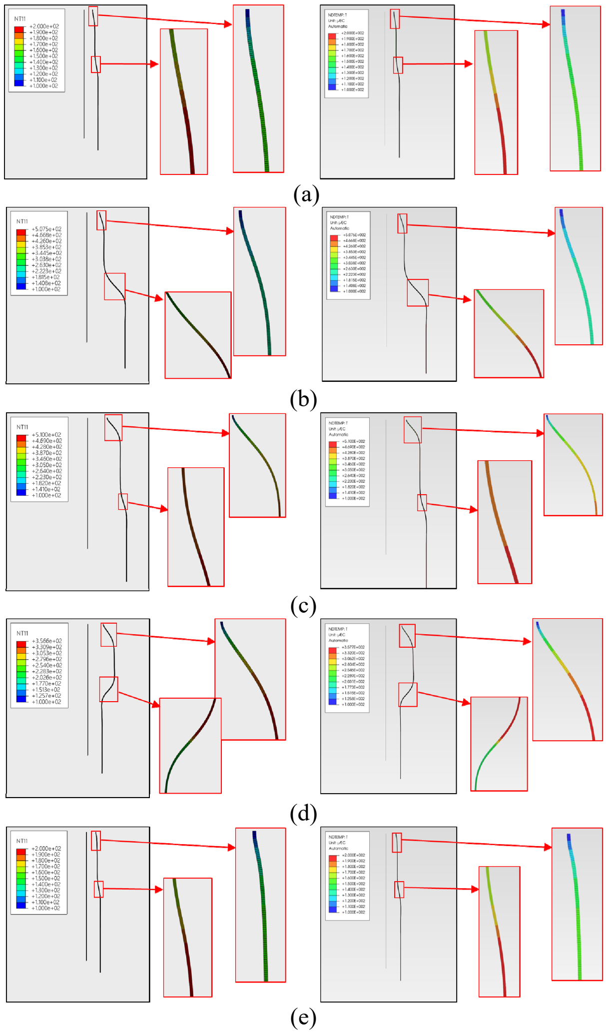

Fig. 3. The temperature distributions and deformed shapes at [(a) initial (b) startup (c) normal operation (d) shutdown and (e) reloading states] (left: Abaqus; right: CalculiX)

Table I: Maximum temperatures of the reactor vessel for each step of Abaqus and CalculiX

Step	Max. Temperature (ºC)		
	Abaqus (Coupled)	CalculiX (Uncoupled)	CalculiX (Coupled)
1	200	200	200
2	507.5	507.6	507.6
3	510	510	510
4	356.6	357.1	357.7
5	200	200	200

In step 5, the condition of step 4 is maintained, and the upper part of the vessel also changes to its original temperature state. As the temperature of the cover gas region also decreases, the overall displacement becomes similar to that of step 1.

The maximum values of radial displacement and stress for the three kinds of analyses are compared and do not show significant differences. In order to check the displacement patterns for the entire region, the radial displacements of the inner surface of the vessel were compared. Since the sequentially and fully coupled analysis results of CalculiX are almost identical, only the fully coupled analysis results of Abaqus and CalculiX are illustrated in Fig. 4. The horizontal axis of Fig. 4 is the vertical coordinate of the inner surface of the reactor vessel when the normal service level is set to 0, and the vertical axis indicates the radial displacement of each point. The displacements calculated from both analyses for all steps agree very well over the entire region, and the validity of the nonlinear plastic analysis using CalculiX can be confirmed.

Figure 5 compares the radial displacement of the reactor vessel in the initial condition and after the end of the 1st and 10th cycles in the CalculiX analysis. It can be seen that the radial displacement slightly increases as the cycle is repeated. The increase in permanent deformation caused by thermal ratcheting, however, cannot be confirmed from the displacement in which elastic and plastic deformations are mixed. In order to quantitatively compare the degree of accumulation of ratcheting deformation, the plastic strain out of the total strain, especially the circumferential plastic strain component, is needed. Unlike Abaqus, however, functions to extract plastic strain components from CalculiX cannot be found, and therefore quantitative evaluation was not performed.

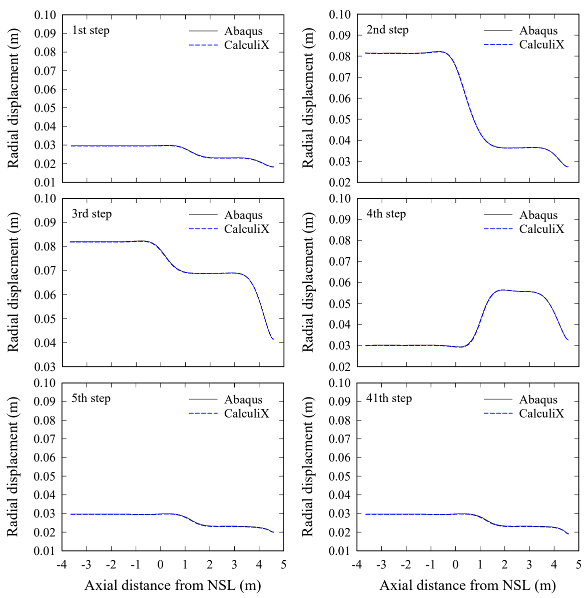

Fig. 4. Comparison of radial displacements from thermal ratcheting analysis of SFR using Abaqus and CalculiX

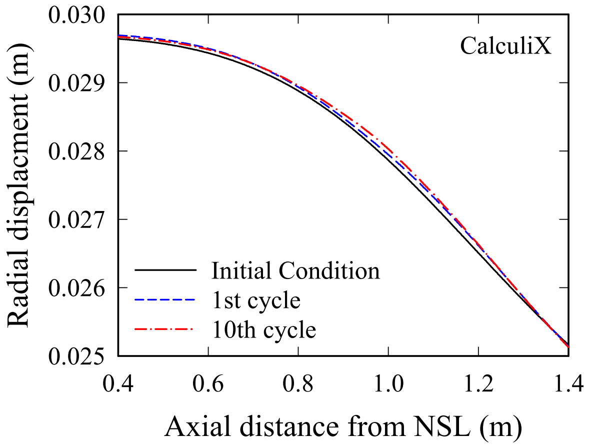

Fig. 5. Comparison of radial displacement by cycle in SFR thermal ratcheting analysis using CalculiX

3. Conclusions

The ratcheting analyses of the SFR reactor vessel were performed using CalculiX, and the validity of the code was verified by comparing them with the analysis of the commercial code, Abaqus. Through the ratcheting analysis, it was confirmed that the Abaqus and CalculiX analysis results agree very well in the elastoplastic analysis. However, it was confirmed that there are some differences in available options and output data. In addition, the analysis time of CalculiX was excessively longer than that of Abaqus. Therefore, when applying CalculiX to structural analysis and FSI analysis, it is necessary to thoroughly review the analysis method, procedure, data extraction, post-processing method, and analysis time in advance.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) grant and National Research Council of Science & Technology (NST) grant funded by the Korean government (MSIT) [grant numbers 2021M2E2A2081061, CAP20032-100].

REFERENCES

[1] D. Kim, J. H. Lee, and J. Hong, Fluid-Solid Interaction Analysis Using a Coupling Library, Trans. KNS Spring Meeting, 2022.

[2] CalculiX CrunchiX User's manual version 2.17, Guido Dhondt, 2020.

[3] Abaqus user manual, SIMULIA User Assistance 2018, Dassault Systemes Simulia Corp.

[4] J. H. Lee, J.-B. Kim, and S.-K. Kim, Racheting Analysis Using Abaqus, KAERI/TR-8183/2020.

[5] S. Okajima, A study on plastic strain accumulation caused by traveling of temperature distribution synchronizing with temperature rise, Mechanical Engineering Journal 3, 15-00574, 2016.