Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2023

Development of Tools for Stress Calculations in Fluid-Structure Interaction Analysis using Open-source Codes: A Case Study on Fatigue Damage Assessment

Jin Haeng Lee [a*], Dehee Kim [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

Fluid-structure interaction (FSI) analysis enables structural analysis with complex boundary conditions such as time-varying pressure and temperature. This eliminates many of the assumptions made during a separate thermal stress analysis and allows for accurate stress calculations, which can be very useful for evaluating fatigue damage in structures. The open-source codes used for these FSI analyses can be used in combination to suit the analysis objectives. From the analysis results, it is necessary to collect the input data used in the evaluation in order to perform an assessment of structural integrity based on ASME Boiler and Pressure Vessel (B&PV) code Section III for nuclear facility components [1]. The inputs include stress components classified by stress linearization for each selected section, as well as material properties, temperature, and, if necessary, time and strain. If multiple loads are acting simultaneously, such as dead weight, pressure, thermal loads, etc., the stresses calculated from the individual analysis of each load are linearized and combined by component. If multiple types of cycles are to be considered, the inputs must be produced for each type of cycle. However, unlike commercial finite element analysis software such as ANSYS and Abaqus, most open-source codes such as CalculiX, a finite element analysis code, and solidDisplacementFoam, a finite volume analysis code provided by OpenFOAM, do not provide their own stress linearization functions, and their data extraction functions are limited. Therefore, in this study, tools that can perform data extraction and stress linearization using these software were developed and validated by applying them to the analysis of a plate subjected to a simple bending load. The developed program was applied to the coupled FSI analysis of a T-junction subjected to thermal striping to evaluate fatigue damage.

2. Stress linearization

2.1 Stress linearization procedure

In order to linearize the calculated stresses calculated from the analysis, it is necessary to know the stress components corresponding to each coordinate on the stress classification line (SCL) defined in the cross section of the structure. In ANSYS, 47 equally spaced interior points are added between the two end points of the line to calculate the stress at each point, and then the classified stress components are calculated through linearization. The stress linearization process depends on whether the analysis is axisymmetric or not. In this study, the stress linearization process of the analysis results was programmed only in the Cartesian coordinate system because the final goal is to apply it to a fully three-dimensional analysis.

In Fig. 1, the path connecting the two end points N_1 and N_2 of the structure cross section is the SCL. The six stress components at the two ends of the SCL and the central point can be divided into membrane stress, bending stress, and peak stress through stress linearization. First, the membrane stress can be calculated through the following integral,

$$\sigma_i^m = \frac{1}{t}\int_{-t/2}^{t/2} \sigma_i \, dx_s \qquad (1)$$

where the subscript i denotes the stress component, the superscript m denotes the membrane stress, t is the distance between the end points of the line, and x_s is the axial coordinate of the path.

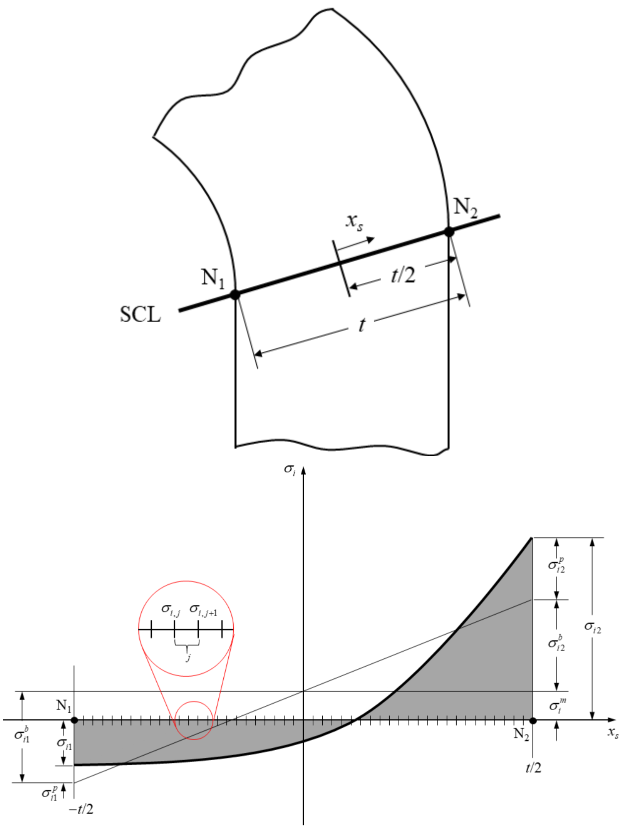

Fig. 1. Coordinates of cross section and typical stress distribution on a stress classification line [2].

In ANSYS [2], Eq. (1) is replaced by the following numerical integration,

$$\sigma_i^m = \frac{1}{n}\left\{\frac{\sigma_{i,1}}{2} + \frac{\sigma_{i,n+1}}{2}\sum_{j=2}^{n}\sigma_{i,j}\right\} \qquad (2)$$

where n is the number of intervals and j is the sequence of each point starting at the N_1. Since ANSYS uses 47 interior points, the number of intervals n is 48.

The moment per unit length for stress component i at the center of the SCL is defined by

$$M_i = \int_{-t/2}^{t/2}\sigma_i x_s \, dx_s \qquad (3)$$

The linearized bending stress σ_{i1}^b at N_1 ($x_s = -t/2$) is given by

$$\sigma_{i1}^b = \frac{M_i(-t/2)}{I} = -\frac{6}{t^2}\int_{-t/2}^{t/2}\sigma_i x_s \, dx_s \qquad (4)$$

Eq. (4) is replaced by the following numerical integration,

$$\sigma_{i1}^b = -\frac{3}{2n}\sum_{j=1}^{n}\left[\left(\sigma_{i,j} + \sigma_{i,j+1}\right)\left(\frac{2j-1}{n} - 1\right)\right]. \qquad (5)$$

The linearized bending stress σ_{i2}^b at $x_s = t/2$ is the same magnitude as σ_{i1}^b, with only the sign reversed. The peak stress is the total stress minus the membrane stress and the bending stress.

In order to calculate the membrane, bending, and peak stresses using the above equations, it is necessary to know the stress components at both end nodes as well as the stress components corresponding to the internal points. In this study, the stresses at each internal point were calculated by linearly interpolating the stresses of the nodes on the SCL.

To perform stress linearization, the coordinates of the nodal points along the SCL and the six stress components are extracted from the analysis results. The extraction process relies on the characteristics of the open-source code, and separate programs were written as needed. In addition to the linearized stresses, the analysis results include the principal stresses, stress intensities, and equivalent stresses.

Since codes such as STEP [3] used for structural integrity assessment based on ASME B&PV Section III are written based on ANSYS analysis results, the output form used here is written to be as similar as possible to the file form output by stress linearization in ANSYS analysis.

2.2 Comparison of the linearized stresses for a simple bending load

To validate the developed stress linearization program, the results of data extraction and stress linearization calculations using Abaqus, ANSYS, and CalculiX were compared for a plate subjected to a simple bending load.

The model used for the analysis assumes a two-dimensional square plate geometry as shown in Figure 2. The length of the plate is 4 m in the transverse (x-axis) direction and 10 m in the longitudinal (y-axis) direction, and the analysis is performed using a plane stress element with a width of 1 m.

An elastic material with Young's modulus of 200 GPa and Poisson's ratio of 0.3 was assumed, and the leftmost node at the bottom of the structure was fixed for both directions of displacement, while the rest of the bottom nodes were fixed for vertical (y-direction) displacement only. The two end nodes at the top of the structure were given a y-direction displacement of -0.01 m on the left and 0.01 m on the right. The stress linearization section was selected in the center of the y-direction. Two types of plane stress elements, 4-node and 8-node elements, were used in the analysis: for the 4-node element, Abaqus and CalculiX used CPS4 element and ANSYS used plane182 element; for the 8-node element, CPS8 and plane183 elements were used, respectively. Table I compares the y-directional positive bending stresses obtained by applying the stress linearization program developed in Calculix to the values calculated by the built-in stress linearization functions in Abaqus and ANSYS when using 4- and 8-point elements, respectively.

The stress linearization results for the three codes were very similar when using 4-node elements. For the 8-node elements, the bending stress in CalculiX obtained by linearizing with the developed code deviated from the value obtained in ANSYS by about 5%. However, the stress obtained in CalculiX was in good agreement with the value in Abaqus.

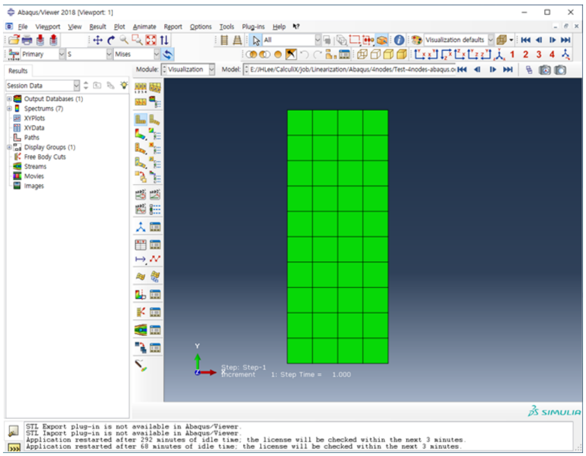

Fig. 2. FE model for the analysis of a two-dimensional flat plate subjected to a simple bending load.

Table I: Comparison of the maximum bending stresses

	CalculiX	Abaqus		ANSYS	
Element type	Max. stress (MPa)	Max. stress (MPa)	Dev. (%)	Max. stress (MPa)	Dev. (%)
4-node	190.8	188.8	-1.05	188.5	-1.21
8-node	167.6	166.2	-0.84	159.8	-4.65

3. Fatigue damage assessment of a T-junction

To demonstrate the applicability of the tools developed, we applied them to the coupled FSI analysis using OpenFOAM and solidDisplacementFoam for a T-junction subjected to thermal striping [4] as shown in Fig. 3, and performed fatigue evaluation based on ASME B&PV code.

The pipe material was assumed to be 316 stainless steel. The analysis results were recorded at 0.1 s intervals for 10 seconds from 5 to 15 seconds. Fig. 4 shows the evolution of the six stress components over time at nodes inside and outside the SCL. Stress linearization was performed using the code developed to input the data into the STEP code. For accurate fatigue damage assessment, the time range of the expected peak and valley points should be determined based on the endurance limit, respectively, as described in the EPRI report [5], and then the peak and valley points should be detected by combining the stress components within this range. However, since this study focuses on examining the applicability of fatigue damage assessment through coupled FSI analysis using open-source code, we used only stresses within a width of 0.1 s from the point where the extreme time is expected for convenience.

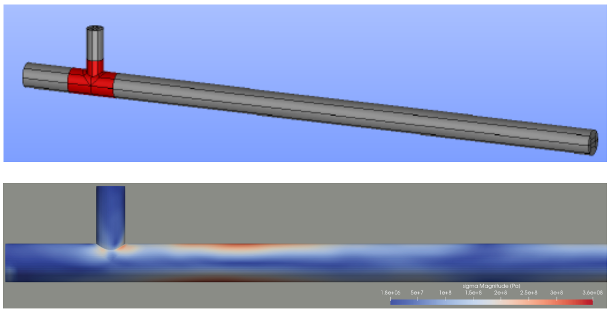

Fig. 3. Coupled FSI analysis model for a T-junction

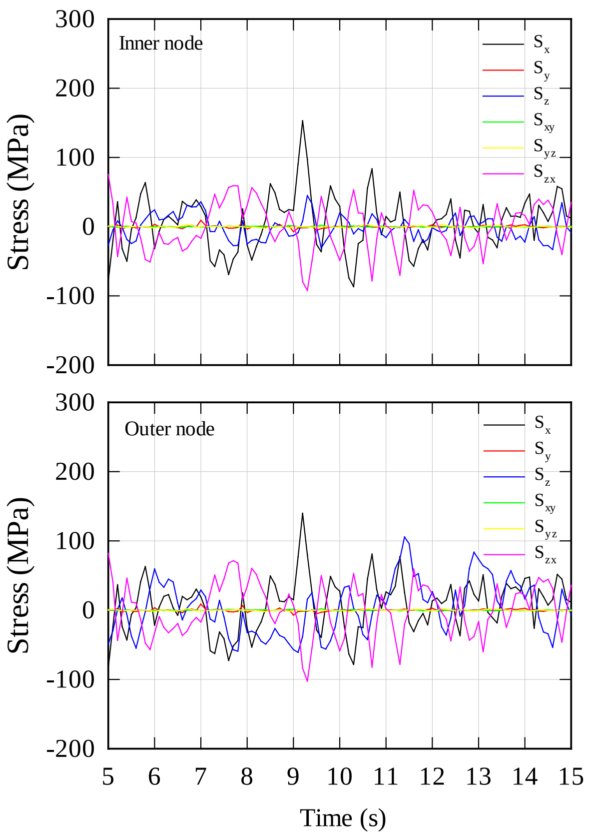

Fig. 4.Stress components over time at inner and outer nodes.

As a result of STEP evaluation, 5 s and 9.2 s were selected as the valley and peak points, respectively, and the alternating stress intensity S_a generated at the outer node of the fatigue pipe was 197 MPa. The endurance limit of austenitic steel is 93.7 MPa, and only one cycle including the selected peak and valley exceeds this limit; the remaining cycles do not affect the fatigue life. Therefore, the estimated number of allowable cycles for the T-junction analysis evaluated by the STEP code is 98261 cycles at S_a = 197 MPa.

3. Conclusions

For coupled FSI analysis and structural integrity evaluation using open-source codes, we developed tools that can perform stress extraction and stress linearization. To validate the developed tools, bending problems were analyzed using the structural analysis codes CalculiX CCX, Abaqus, and ANSYS, and the linearized stress values were compared to verify the similarity of the linearized stresses for different element types. For the T-junction benchmark problem with thermal striping, the stress data obtained through coupled FSI analysis using OpenFOAM and solidDisplacementFoam were evaluated for fatigue damage after stress linearization using the developed tools, confirming that they can be efficiently used for structural integrity evaluation.

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, CAP20033-100].

REFERENCES

[1] ASME Boiler and Pressure Vessel Code, Section III, The American Society of Mechanical Engineers, 2017.

[2] ANSYS Mechanical User Manual. 2020.

[3] N. H. Kim, J. B. Kim and S. K. Kim, Development of a Structural Integrity Evaluation Program for Elevated Temperature Service According to ASME Code, Nucl. Eng. Tech. 53, 2021.

[4] Report of the OECD/NEA-Vattenfall T-Junction Benchmark Exercise, NEA/CSNI/R(2011)5, OECD, 2011.

[5] S. Chu, Stress-Based Fatigue Monitoring: Methodology for Fatigue Monitoring of Class 1 Nuclear Components in a Reactor Water Environment, EPRI Technical Report, 2011.