Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, November 2-3, 2006

Finite Element Analysis for Feed-Water Nozzle of PWR Steam Generator by stress analysis code 3G

Jin H. Choi, S. N. Choi
Korea Electric Power Research Institute, 103-16, Munji-dong, Yuseong-gu, Daejeon, 305-380

1. Introduction

The stress analysis for feed water nozzle of PWR steam generator is a complicated process in that there are both advanced theoretical knowledge and complex programming works due to its intricate geometry. In case of using the ordinary analysis code, a lot of times and efforts are needed because the modeling, the setting of the load condition and the boundary condition, and the meshing for 3-dimensional structure are so complicated. Axi-symmetric 2-dimensional analysis that approximates plane-symmetric 3-dimensional analysis has been commonly performed in the process of the analysis of nozzles. In this work, the 3-dimensional structure analysis and the 2-dimensional structure analysis are performed by using the stress analysis code, 3G which can easily perform the modeling, the setting of the load and boundary conditions, and the meshing. The results of the 3-dimensional and 2-dimensional analyses are compared with each other. The result of the 2-dimensional analysis is compared with that of the 2-dimensinal axi-symmetric analysis by ANSYS as well. The free(triangular) meshes and mapped(quadratic) meshes are also compared respectively. The maximum principal stresses are used in the comparison of results.

2. 2-Dimensional Stress Analysis

2.1 2-Dimensional Stress Analysis by ANSYS

2.1.1 Finite Element Model

The preprocessor module of ANSYS is used to create the 2-dimensional axi-symmetric geometry model for the stress analysis using ANSYS[1]. The materials properties of the feed water nozzle in the steam generator are presented in Table1[2]. The direction of the pipe connected to the nozzle is an symmetric axis. The length of the pipe is extended to minimize the effects of the boundary. L is an extended length of the pipe. R is a mean radius of the pipe. T is a thickness of the pipe.

$$L \geq 2.5\sqrt{RT}$$

The quadratic elements with 8 nodes and ANSYS version 9.0 are used in this analysis.

2.1.2 Load and Boundary condition and results

The internal pressure of 1000psi is applied to the inner surface of the steam generator shell , nozzle and pipe.

Materials	Young's modulus (psi)	Poisson ratio	Density (lb/in^3)
SA508 Class3	27.6E6	0.3	0.283
SB564 Alloy690	30.1E6	0.3	0.283
SA234	29.5E6	0.3	0.283

Table 1. Materials Properties

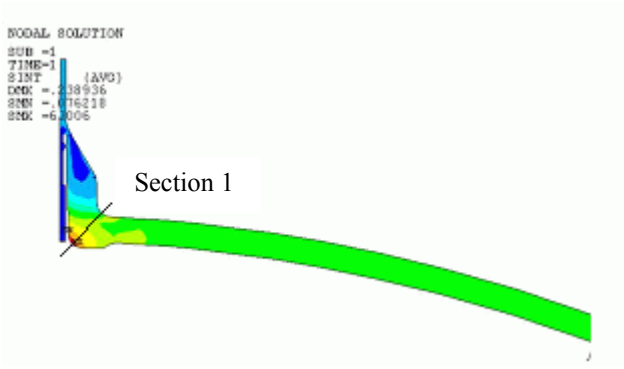

Figure 1. Diagram of the maximum principal stress in 2-dimensional analysis using ANSYS

The axial stress is also applied to the extended directional surface of the pipe.

$$Axial\ stress = PR^2/(R_2^2-R_1^2)$$

P is an internal pressure. R_1 is a inside radius and R_2 is a outside radius. The results of the stress analysis are presented in Figure 1. They show a distribution of the maximum principal stresses. Afterwards, the maximum principal stresses on the Section 1 will be compared.

2.2 2-Dimensional stress analysis by 3G

The 3-Dimensional CAD (Computer Aided Design) software is used to create the geometry model for analysis.

The same load and boundary conditions are applied as the analysis by ANSYS. But the quadratic element is not supported to enhance conveniences of the analysis in 3G so that the triangular elements with 6 nodes are used. While all load and boundary conditions are applied to the line in ANSYS model, they are applied to the surface in 3G model because the 2-dimensional model in 3G is extended to the virtual 3-dimensional model by revolving around the axis of symmetry.

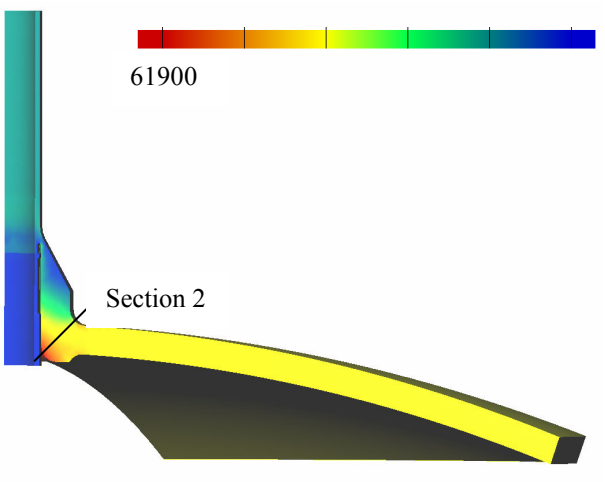

Figure 2. Diagram of the maximum principal stress in 2-dimensional analysis using 3G

A quarter of the full model is analyzed considering the symmetry condition. The results are presented in Figure 2. The maximum principal stresses on the Section 2 will be also compared.

3. 3-Dimensional Stress Analysis

3.1 Finite Element Model

The 3-Dimensional CAD software is also used to create a 3-dimensional geometry model for analysis. The 3-dimensional geometry model is so complicated, which is a reason why an axi-symmetric models have been used. The length of the pipe is extended and that of the shell is also extended. The tetrahedral elements with 10 nodes are used in the analysis. The advantage of 3G is that we can easily create 3-dimensional finite element model.

3.2 Load and Boundary condition and results

The internal pressure of 1000psi is applied to the inner surface of the shell, nozzle and pipe. The axial stress is applied to the upper surface of the pipe and the shell. The boundary condition is applied to the plane symmetric surfaces.

A quarter of the full model is also analyzed considering the symmetry condition. The results are presented in Figure 3. The maximum principal stresses on the Section 3 will be also compared. The inner edges of all three nozzles show the maximum value of the maximum principal stress. Therefore the value of that regions are compared one another.

4. Conclusion

The profiles of the maximum principal stress on Section 1, 2 and 3 are presented in Figure 4. The results of the 2-dimensional analysis in ANSYS and 3G show that we can obtain similar distributions of the maximum

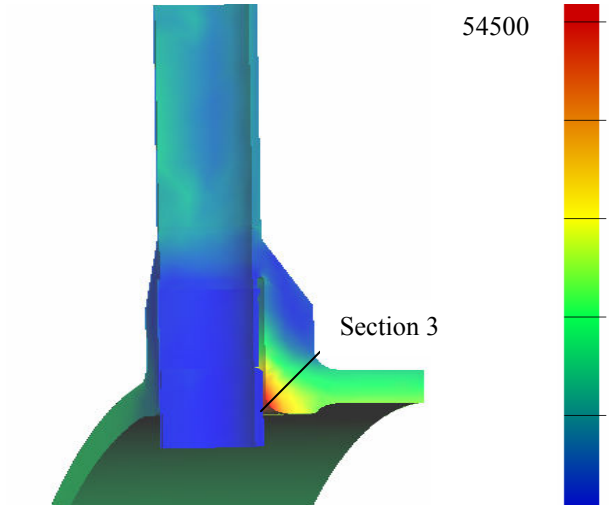

Figure 3. Diagram of the maximum principal stress in 3-dimensional analysis using 3G

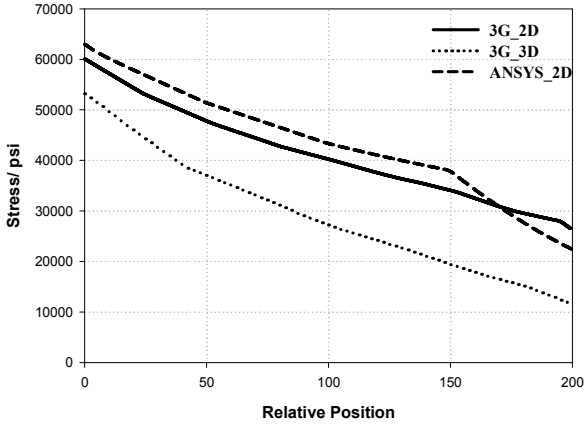

Figure 4. Profiles of the maximum principal stress on Section 1, 2 and 3

principal stresses and the acceptable profiles of the Section 1 and 2. Therefore It can be said that the 2-dimensional stress analysis using 3G is reliable in this works. From the results of the 2-dimensional analysis and 3-dimensional analysis, the 2-dimensional analysis is more conservative like other analysis codes. Because of these conveniences and conservativeness, the 2-dimensional stress analyses have been performed in Nuclear Power Plants.

REFERENCES

[1] ANSYS users manual. Version 9.0

[2] ASME Boiler and Pressure Vessel Code Section III, Division 1, “Rules for Construction of Nuclear Power Plant Components”, 1992.

[3] S.N. Choi, I.C. Lee, and K.S. Jang “Three-Dimensional Versus Axisymmetric Finite-Element Analysis of Reactor Vessel Safety injection Nozzle,” Korean Society of Mechanical Engineering spring conference A pp. 844~849, 1998.