Finite Element Analysis

By Stephen Lam, Senior Engineer, Boilers and Pressure Vessels Safety Program

According to ASME Section VIII DIV 1, U-2(g), when rules for the details of design and construction are not given in the Code, the Manufacturer, subject to the acceptance of the Inspector, shall provide design and construction details which will be as safe as those provided by the rules of the Code.

In this situation, Finite Element Analysis (FEA) is often chosen as the tool as it is very powerful and relatively inexpensive. Unfortunately, many designers treat FEA programs as automated tools for design and ignore the fact that FEA can render solutions that are seriously flawed if conducted by an inexperienced person. It is therefore very important that FEA be performed by someone who is experienced in this analysis, the particular software used and the field of the design.

As FEA is becoming more common, TSSA is receiving more and more submissions for registration with designs supported by FEA. It should, however, be understood that FEA may be used only for designs which are not covered by the rules of the Code. As FEA requires extensive knowledge and experience, TSSA requires the FEA report be certified by a professional engineer. The FEA report shall include the following:

Summary

The summary shall briefly describe the purpose of the FEA, the justification of using FEA, the FEA model, the results and the conclusion.

Introduction

The introduction shall include the assumptions used to perform the design, the name and version of the software used, and the type of FEA analysis (elastic, plastic, small deformations, large deformations, etc.).

FEA Model

This section must describe the model completely and include the following.

1. Type of finite element model used (solids, shells, beams, 2D, etc.).
2. Material models utilized for all required physical properties and strength parameters.
3. Geometry with dimensions.
4. Description of loading and boundary conditions such as loads, restraints and supports, etc. for all load cases (these must be shown in the figures).
5. Explanation of partial models, if any, such as 'due to symmetry'.
6. Description of the finite element model including the division of elements.
7. Type of elements used (triangular, square, rectangular, etc.), mesh size and the number of elements.
8. Number of Degrees of Freedom of the model.
9. The element order (must be at least second order).
10. The turn angle of the elements in the mesh.
11. The types of elements used (h-elements, p-elements).
12. The method used to estimate the error of the results and the maximum percentage of combined error.

Results

All results shall be included here. The figures must be in colour and clear (enlarged sections of some figures may be required). In some cases, the figures should also be submitted in electronic form to facilitate review. Explanations and discussions on each figure must be provided. Figures showing the following shall be provided.

1. The mesh of the model.
2. The displacement/deformation.
3. The stresses.
4. Convergence, and/or plot with element stress and a comparison of nodal (average) stress vs. element (non-averaged) stress.

Conclusion

The conclusion shall include the acceptance criteria of the Code of construction. The results of the FEA shall be compared with the Code requirements. Any verification techniques used to check the model and the results should also be included.