Calculation and Comparison of Fracture Parameter of 3-D Thin-walled Structure in ANSYS

Jinsan JU

Dept.of Civil Engineering, National University of Ireland, Galway

Abstract:

Due to the complication of stress distribution around the crack tips, it is hard to obtain the fracture parameters (Stress intensity factor, J-integral factor) in a 3-dimention complex model. In order to get the accurate result of SIF of 3-D thin-walled structure, this paper describes a calculation procedure of 3-dimention thin-walled models. It uses ANSYS to create model with singular elements around the crack tips and obtain SIF accurately. An example model of one part of barrel with crack is calculated. Through results comparison with that of theoretical formula and results of ABAQUS, it shows that the SIF calculation of ANSYS is accurate. To make the calculation procedure high efficient, this paper made a program for the barrel model using APDL to automate the model creation and analysis procedure, another program is also made to transfer model including boundary conditions from ANSYS to ABAQUS automatically.

Introduction

The fracture mechanics calculation of 3-dimention models is difficult. Some kinds of methods to simplify the model or calculation procedure were carried out (Reference 1). Basically, the fracture parameters can be obtained from 2-dimention local models at last from these methods but it needs a very refined mesh around the crack tip and needs much efforts (Reference 2). So, it is necessary to find an easy way to get the fracture parameters directly in 3-D model with rather coarse mesh around the crack tip.

Calculation procedure in ANSYS

In the study of fracture mechanics interest is often focused on the singularity point where quantities such as stress become (mathematically, but not physically) infinite. Near such singularities normal, polynomial-based finite element approximations perform badly. Therefore, the singular element should be used around the singularity point (crack tips). Along the singular element edges the derivatives $\partial u / \partial x$ (strain) very as $1/\sqrt{r}$ where r is the distance from the corner node at which the singularity develops (Reference 3).

In linear elastic problems, stress intensity factor (SIF) is a very important parameter of fracture mechanics from which the singular stress and strain around crack tip can be obtained. In this paper, ANSYS, a kind of FEM software with general analysis function in engineering and research, is used to create a procedure for calculating the SIF of 3-D thin-walled model such as a part of barrel with crack. At the same time, SIF are calculated by ABAQUS as well for comparison. The same barrel with another mesh for ABAQUS is created in ANSYS and transferred from ANSYS to ABAQUS automatically using APDL in ANSYS.

Following are the procedures of calculation in ANSYS:

1. Create model and use singular elements around the crack tips;
2. Solve;
3. Calculate SIF in postprocessor.

Creating model in ANSYS

For elastic problems, the displacements near the crack tip (or crack front) vary as $1/\sqrt{r}$, where r is the distance from the crack tip. The stresses and strains are singular at the crack tip, varying as $1/\sqrt{r}$. To pick up the singularity in the strain, the crack faces should be coincident, and the elements around the crack tip (or crack front) should be quadratic, with the mid-side nodes placed at the quarter points. Such elements are called singular elements (Reference 4). In this paper the element shell93 is used for the 3-D thin-walled model in ANSYS.

In 3-D thin-walled structures, the barrel is the most typical one. A part of barrel is analyzed here for SIF calculation. The model shown in figure 1 is the model of barrel panel under pressure 0.0593 N/mm^2, radius=2820 mm, length of arc is 740 mm, length of straight line is 1066.8 mm, thickness of panel is 1.8 mm, elastic modulus 73100 N/mm^2. The six freedom of each node on four sides are all constrained. A program is made in ANSYS by APDL to make this procedure of barrel calculation automatic including creating model, applying boundary conditions, solving and post processing.

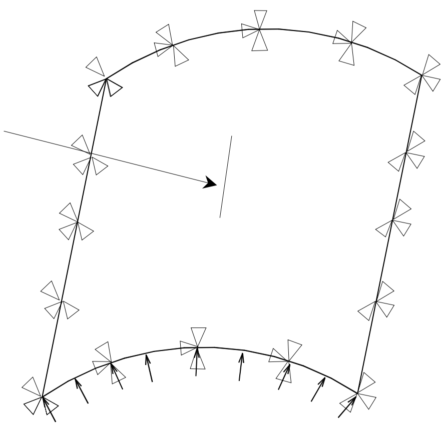

Figure 1. Barrel model

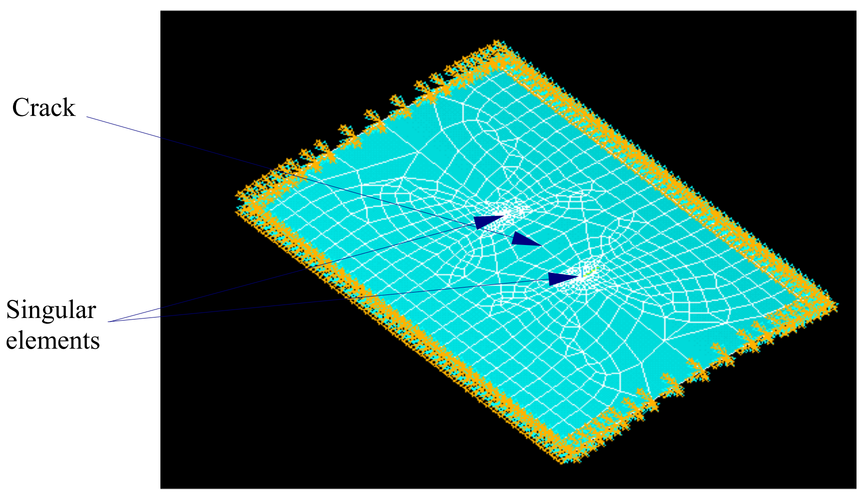

Figure 2. Barrel model in ANSYS

Creating model in ABAQUS

The same barrel model is also created in ABAQUS in order to calculate the other result for comparison with that of ANSYS. There are two keyholes at crack tips and the focused mesh is used there, which is the special point for ABAQUS calculation (Reference 5). Therefore the meshed model in ANSYS is different from that of ABAQUS (shown in figure 3). Because ABAQUS's preprocessor is too poor to create this model, ANSYS's excellent preprocessor is used for creating model for ABAQUS as well.

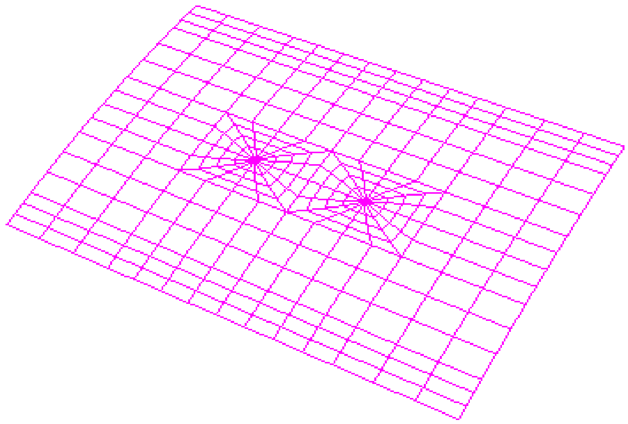

Figure 3. Barrel model in ABAQUS

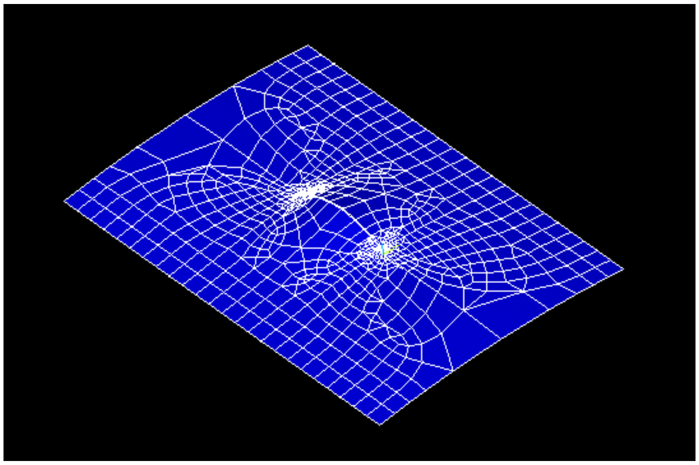

Figure 4. Deformation of barrel model under inner pressure in ANSYS

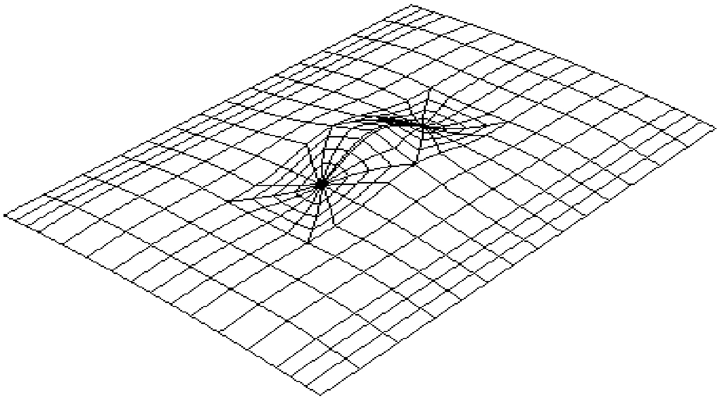

Figure 5. Deformation of barrel model under inner pressure in ABAQUS

Another program is also made for the transfer of model from ANSYS to ABAQUS. It can output all the information in ANSYS to an input file with ABAQUS format. This file has all the information of the model including material character, nodes, elements and boundary conditions. This program is a general purpose one. It can transfer all the models using shell element from ANSYS to ABAQUS.

Calculation results analysis

There are three kinds of results of the stress intensity factors calculated shown in figure 6. The horizontal axis is the half crack length and the vertical axis is the SIF. From up to down, the first curve is the result from stress intensity factor formula (1) of barrel (Reference 6).

$$K_I = \frac{PD}{2h}\sqrt{\pi a}\left[1 + 3.22\frac{a^2}{Dh}\right]^{1/2} \tag{1}$$

where K_I is the stress intensity factor; D is the diameter of barrel; h is the thickness; a is the half length of crack; P is the inner pressure. The second result is of the model of figure 3 in ABAQUS; the third one is the result of the model in figure 2.

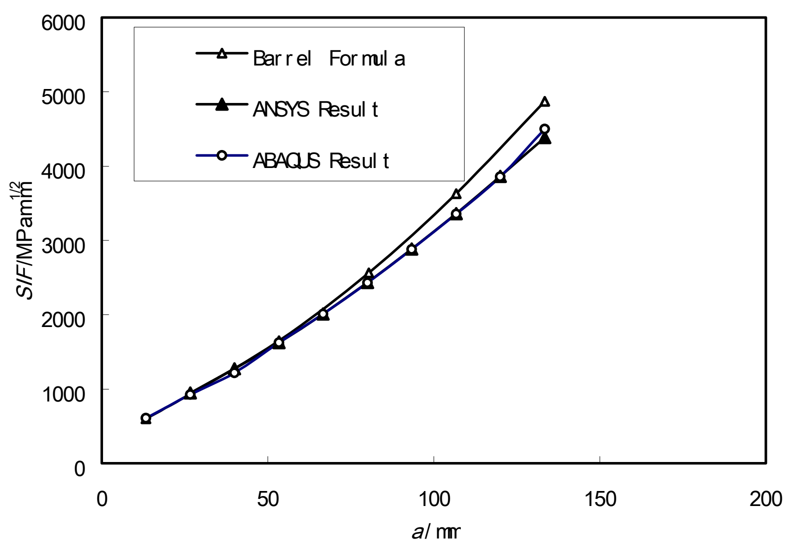

Figure 6. SIF results of barrel model

From the results shown in Figure 6, it can be seen calculation results of barrel from ANSYS and ABAQUS are very similar to theoretical results for different crack lengths. It means the calculation results are reasonable. At the same time, the results from ANSYS are almost the same as that of ABAQUS. It means that the calculation procedure in ANSYS is feasible.

When the deformation of the models in ANSYS shown in figure 4 is checked and compared with that of ABAQUS in figure 5, it can be seen that the deformation is very close to each other. It means that the calculation is accurate although there are different model meshes in ANSYS and ABAQUS. The model with focused mesh around the crack tips in ABAQUS is much complex than that in ANSYS while the singular elements are created automatically around the crack tips. It needs much more efforts to create model in ABAQUS than that in ANSYS but the same calculation results can be obtained from them. It means that this method in ANSYS is high efficient. It had been used in much more complex structures by far.

In general, the procedure of calculating SIF in ANSYS is accurate and high efficient.

Conclusion

This paper investigated the procedure of SIF calculation in ANSYS of 3-D thin-walled structure. A part of barrel model is calculated using this procedure. The SIF of the same barrel is calculated in ABAQUS as well. These two kinds of SIF results are compared with theoretical results. The comparison shows that calculation result is accurate and the procedure is high efficient in ANSYS. At the same time, two programs are made for the automation calculation in ANSYS and ABAQUS.

References

1) T.L. Anderson, Fracture Mechanics Fundamentals and Applications. Florida, CRC Press, 1995
2) S.N. Atluri, Structural Integrity and Durability, Tech Science Press. 1997
3) O.C Zienkiewicz, The Finite Element Method. 3rd Edition. London, McGraw-Hill. 1977
4) X Zhang, N Hasebe, “Interactions Between Rectilinear and Circumferential Cracks”, Engineering Fracture Mechanics, 46(2): 311-328.1998
5) Hibbitt, Karlsson & Sorensen, Inc. ABAQUS/Standard User’s Manul.1969-1997.2002
6) M.F Kanninen, C.H Popelar, Advanced Fracture Mechanics. New York, Oxford University Press, 16-18.1985