RESEARCH ARTICLE OPEN ACCESS

Summary of fracture mechanics problems analysis method in ABAQUS

Duan Hongjun*, Guo Guixiang**
*(Department of Bridge and tunnel Engineering, Chongqing Jiaotong University, Chongqing 400074, China)
**(Materials Engineering, Chongqing Jiaotong University, Chongqing 400074, China)

ABSTRACT
Fracture mechanics is the study of the strength of the materials or structures with crack and crack propagation regularity of a discipline. There are a lot of analysis function of ABAQUS, including fracture analysis. ABAQUS is very easy to use and easy to establish a model of the complicated problem. In order to effectively study of strong discontinuity problems such as crack, provides two methods of simulating the problem of cracks of ABAQUS. This paper describes the two methods respectively, and compare two methods.

Key words-ABAQUS、fracture mechanics、XFEM、CFE

I. INTRODUCTION

Fracture mechanics is the study of the strength of the material or structure with crack and Law of crack propagation. Fracture mechanics can be divided into two parts. One part is linear elastic fracture mechanics. Another part is elastoplastic fracture mechanics.

The research object of linear elastic fracture mechanics is line elastomer with crack，it is based on linear elastic theory. For a variety of complex fracture mode, it is also broken down into three basic fracture type. These three basic type of fracture is Ⅰ-type fracture , Ⅱ-type fracture and Ⅲ-type fracture.

（1）Ⅰ-type fracture belongs to open-type fracture. The crack that is perpendicular to the direction of pull and through the thickness on the tensile plate belongs to Ⅰ-type fracture. In addition, the rupture which the longitudinal crack in the cliff of the long cylindrical container or the pipe forms under the action of internal pressure also belongs to Ⅰ-type fracture.

（2）Ⅱ-type fracture belongs to the sliding mode fracture. The crack caused by the crack along the tangent direction belongs to Ⅱ-type fracture. In addition, the rupture which the circumferential crack the cliff of the pipe on the torsional thin-walled circular tube forms under the effect of torque also belongs to Ⅱ-type fracture.

（3）Ⅲ-type fracture belongs to tear-shaped fracture. The crack which annular groove cutting or surface circumferential crack on circular shaft or the circular sample forms when the torsional effect of the circular shaft belongs to Ⅲ-type fracture.

For the wide range of yield fracture problems and all yield fracture problems that have a large plastic zone at the end of the crack, linear elastic fracture theory is no longer applicable, the elastic-plastic fracture theory must be used to analyze. At present, there are many kinds of methods of searching the elastic-plastic fracture, but the COD method and j-integral method are the most common.

However,fracture mechanics is a branch of solid mechanics. There are two kinds of classic discontinuous in solid mechanics problems. One kind is discontinuous problem caused by material properties of mutation, this kind of problem is weakly discontinuous problem, on behalf of the problem is the double material problem and mixed problem, its complexity is caused by discontinuous physical interface strain. Another kind is internal object geometric discontinuity problems caused by the mutation, this kind of problem is the strong discontinuity problems, on behalf of the problem is that the crack problem, its complexity by geometric interface displacement discontinuity and end singularity.

To solve the second type of discontinuity, the crack problem, ABAQUS provides several methods..

II. SEPARATE NARRATION

Numerical methods (such as finite element, boundary element, Element-free Method etc.) has been dealing with crack discontinuous problems. Finite element method possesses the advantages that other numerical method can't have, it possesses many advantages, such as being suitable for arbitrary geometry and arbitrary boundary conditions, various nonlinear problems (geometric nonlinearity, material nonlinearity and contact problem), and easy programming, thus it becomes a major means of numerical analysis of discontinuous problems such as crack.

In order to effectively study strong discontinuity problems such as crack, ABAQUS provides two methods to simulate the crack problem. One is based on the conventional finite element method (CFEM) on the research of the crack problem, this method requires the user to establish the model grid that i is consistent with the actual situation of crack. Another method is based on extended finite element method (XFEM) for crack problem research, there is no need for the user to establish the model grid that is consistent with the actual situation of crack.

1.1 THE CONVENTIONAL FINITE ELEMENT METHOD（CFEM）

Conventional finite element method (CFEM) uses a continuous function as the interpolation function, it requires that the interpolation function is continuous and the quality of the material is also continuous. In dealing with the strong discontinuity of crack, the crack surface must be set to element's edge, The crack tip must be set to element's node, in the high stress area of the crack tip need high grid density，and when simulating crack formation it needs to divide the grid again.

1.1.1 THE METHODS OF DEFINING SEAM

With the methods of defining seam to simulate the crack cracking problems, we need to set up the expansion of the crack path. As the load applied, crack is spread along the seam that we define. The methods of defining seam can be used to simulate the singularity of crack tip. Definition of seam of this method, to calculate the J integral and stress intensity factor.

1.1.2 USING THE DEBOND COMMAND

When using debond command is used to simulate crack initiation, We need the specified crack initiation path defined on a set, this method is very simple, but the actual application is quite limited.

1.1.3 USING COHESIVE ELEMENT

When using cohesive element is used to simulate crack initiation, we need to define the damage initiation criterion and rule of evolution. In the post-processing ,we can observe the crack extension through the display group. the actual application is not only not only use widely, but also can be implemented on the platform of ABAQUS simulation crack propagation.

1.2 THE EXTENDED FINITE ELEMENT METHOD (XFEM)

The extended finite element method (XFEM) not only doesn't need a model of the grid that is consistent with the actual crack problems, but also can take place inside the crack. That is to say, actual cracking is independent of that we divided artificially grid , this is more consistent with the actual crack situation.

The function of the newly launched in ABAQUS 6.9 is the extended finite element method (XFEM). It uses the extended finite element method (XFEM). Division of the grid has nothing to do with internal structure geometrical or physical interface, So in the crack tip mesh of high stress and deformation, high precision is no longer a problem, and at the simulation of crack initiation we no longer have to divide the grid subdivision again. It greatly reduces the user's operation and improves the accuracy.

When using ABAQUS simulation grid craze, users don't need to define the extension path. We simply set the stability of the control such as the criterion of crack initiation, crack initiation and crack parameters ,to realize the crack extension.

III. SYNOPSIS

Based on the platform of ABAQUS, Extended finite element method (XFEM) inherits all the advantages of the conventional finite element method (CFEM), In addition there are other different advantages of the conventional finite element method (CFEM), Firstly, division of the grid has nothing to do with internal structure geometrical or physical interface. Secondly at the stress and deformation contour clusters such as crack tip ,we no longer need precision of meshing.Thirdly, when we simulate the crack initiation, we also don't have to divide grid subdivision again. So relative to the conventional finite element method（CFEM），extended finite element method (XFEM) is especially effective on the strong discontinuity problems such as crack growth problems.

Removing the method that we mentioned above, we can also realize through other ways,such assecondary development, The boundary conditions changing with analysis of step, model symmetry and so on.

In a word, ABAQUS provides a lot of methods to deal with crack,the function is also very powerful. If we can get more accurate data, simulating the crack problem is reference value.

References

[1] Abaqus Analysis User's Manual
[2] Abaqus Theory Manual
[3] I.Babuska and J. Melenk, Int. J. Numer. Meth. Engng(1997),40:727-758
[4] Kuang Zhenbang. The crack tip field. Xi 'an jiaotonguniversity press,2002
[5] T.Belytschko and T.Black,Int.J.Numer.Meth.Engng (1999),45:601-620
[6] China aviation institute. Stress intensity factor

handbook. Science press,1993
[7] A. Hansbo and P. Hansbo, Comp. Meth. Appl. Mech. Engng (2004),193:3523-3540
[8] J. H. Song. P .M. A. Arcias and T. Belytschko. Int.J. Number. Meth. Engng(2006),67:868-893
[9] Belytschko T,Lu Y Y,Gu L.F.lement free Galcrkin methods [J].Tnternational Journal for Nmerical Methods in Enginccring,1994,37:229-256.
[10] Sukumar N,Chopp D L,Moes N,Bclytschko T.Modelling holes and inclusions by level sets in the cxtcnded finitc element method [J].Computer Methods in Applied Mcchanics and Engineering, 2001, 190: 6183-6200.
[11] Bazant Z P. Crack band model for fracture of gconmatcrals [A]. In: Proceedings of the 4th International Conference on Numerical Methods in Geomechanics [C]. University of Alberta, Edmonton, 1982,3:1137-1152
[12] Daux C,Moes N,Dolbow J,Sukumar N,Bclytschko T.Arbitrary branched and intersccting cracks with the cxtcnded finitc element method [J].Intermation Journal for Numerical Methods in Engineering,2000,48:1741-1760.
[13] Jirasek M. Comparative study on finitc elements with cmbedded discontinuities [J]. Comput.Methods Appl.Meeh.Eng.2000,188:307-330.