Three-Dimensional Static and Dynamic Stress Intensity Factor Computations Using ANSYS

X. M. Jia
Chongqing Communications Research & Design Institute,
Chongqing China
F. Dai Q. Z. Wang
Department of Civil Eng. & Applied Mechanics,
Sichuan University, Chengdu China

Abstract

In three-dimensional computation for linear elastic fracture mechanics, how to simulate the stress singularity near the crack tip has been a difficult and important point. The so-called quarter-point element is often used to model the stress and displacement field near the crack tip. However, ANSYS only provides automatic meshing capability for two-dimensional problems. It cannot directly generate crack elements for three-dimensional models. At the crack tip region we generate the quarter-point element manually to model the correct singularity of the stresses near the crack tip, thus making the computation for three-dimensional crack problem possible. Manual generation of elements may be tedious for a large and complex model. In this paper, two methods are presented to compute three-dimensional Stress Intensity Factors (SIFs). Firstly, submodel and partial crack submodel methods are adopted to compute the SIFs. Manual generation is only needed for submodel region, which is of much reduced size, thus manual generation is feasible. Secondly, mesh200 element can be used to mesh the area with two dimensional singular elements, and then sweep this area through certain coordinate system to establish three-dimensional crack elements. Finally, three static and dynamic crack examples are given to prove the correctness and ability of these methods. The accuracy of these methods is guaranteed compared with other literature. These two methods are easy to handle and extend the ability of ANSYS in computing three-dimensional SIF.

Introduction

Fracture Mechanics provides a theory background for material and structures containing cracks and faults, and stress intensity factor (SIF) is a key parameter in crack analysis. SIF plays a dominate role because it indicates the singular intensity of linear elastic crack field (stress and strain).

Because of the importance of SIF, Its solutions have been paid very high attention since the beginning of fracture mechanics. Exact solutions to these problems are limited to a few special configurations; e.g., elliptical cracks embedded in very large bodies, we can look it up in SIF manuals. For complicated configurations, such as the intersection of an area crack with a free surface is referred to as a surface flow, where exact solutions are both difficult to obtain and generally not available. Side by side with the difficulty of the problem, there is the unfortunate fact that such surface flaw is the most commonly encountered defect in many engineering structures.

Till now, many methods are adopted to compute SIF, such as finite element method, boundary element method and finite difference method etc. Finite element method is the most popular tool in computing SIF. In three-dimensional computation for linear elastic fracture mechanics, how to simulate the stress singularity of $r^{-1/2}$ near the crack tip (r is the normal distance to the crack tip) has been a difficult and important point. The so-called quarter-point element is often used to model the stress and displacement field near the crack tip; however, its application in the general-purpose finite element software is still difficult. For example, ANSYS only provides automatic meshing capability for two-dimensional problems; it cannot directly generate crack elements for three-dimensional models. At the crack tip region we generate the quarter-point element manually to model the correct singularity of the stresses near the crack tip, thus making the computation for three-dimensional crack problems possible. Manual generation of elements (including crack elements) for the whole model may be tedious for a large and complex model.

In this paper, two methods are presented to compute three-dimensional SIFs. Firstly, submodel and partial crack submodel methods are adopted to compute the SIFs. Manual generation is only needed for submodel region, which is of much reduced size, thus manual generation is feasible. Secondly, mesh200 element can be used to mesh the area with two dimensional singular elements, and then sweep this area through certain coordinate system to establish three-dimensional crack elements. Finally, three static and dynamic crack examples are given to prove the correctness and ability of these methods. The accuracy of these methods is guaranteed compared with other literature.

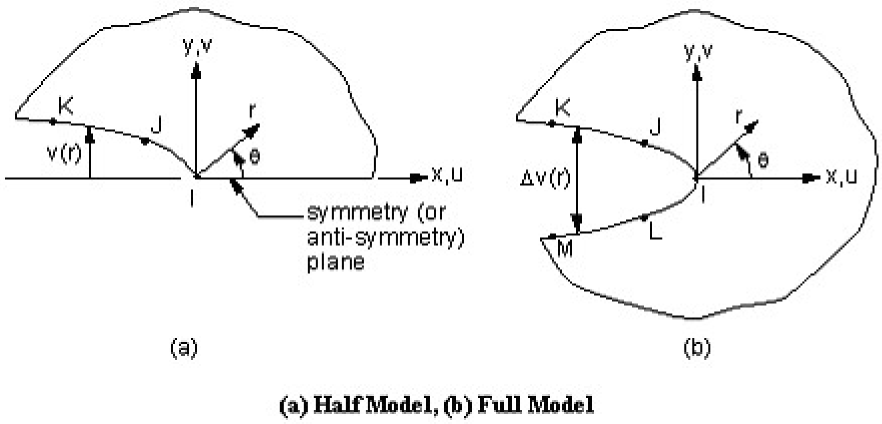


Figure 1. Nodes used for the approximate crack-tip displacements

Procedure

Formulae in computing static and dynamic SIF

As an example, we illustrate mode-I (opening mode) crack and give the formulae to compute the SIF.

For half model:

$$K_I = \sqrt{2\pi}\,\frac{2G}{1+\kappa}\frac{v}{\sqrt{r}}$$

For full model:

$$K_I = \sqrt{2\pi}\,\frac{G}{1+\kappa}\frac{\Delta v}{\sqrt{r}}$$

Where

K_I =

The stress intensity factor of mode-I crack;

G =

Shear modulus;

κ =

$3-4\mu$ if plane strain or axisymmetric; $(3-\mu)/(1+\mu)$ if plane stress; where μ is Poisson's ratio;

$v, \Delta v =$

Displacements in a local coordinate system for half and full model;

$r =$

Coordinates in a local coordinate system.

In dynamic fracture mechanics, dynamic SIF is a function of time t. The formulae are similar with static ones.

For half model:

$$K_I^{dyn}(t) = \sqrt{2\pi}\,\frac{2G}{1+\kappa}\frac{v(t)}{\sqrt{r}}$$

For full model:

$$K_I^{dyn}(t) = \sqrt{2\pi}\,\frac{G}{1+\kappa}\frac{\Delta v(t)}{\sqrt{r}}$$

Where

$K_I^{dyn}(t) =$

Dynamic stress intensity factor of Mode-I cracks; it is a function of time t;

$v(t), \Delta v(t) =$

Dynamic displacements in a local coordinate system for half and full model, They are also the functions of time t.

In dynamic fracture analysis, we use Newmark time integration method [1] to solve kinetics equations for implicit transient analyses, and then we get $u(t)$ information for computing dynamic SIF.

We can use displacement extrapolation just as ANSYS program does in KCALC command; they almost get identical results in computing SIF.

Submodel method and partial crack submodel method

The submodel method is a technique of finite element analysis to obtain a more accurate numerical value for the specific region in the analyzed model with high efficiency; the method is also called cut-boundary displacement method or the specified boundary displacement method. It allows separating a local part of the model from the remaining part, and re-analyzing the submodel with renewed fine mesh. The cut-boundary of the submodel is prescribed by the displacement calculated by the whole model. The submodel method is based on the Saint-Venant's principle, that is, if the actually distributed boundary traction is replaced by the statically equivalent boundary condition, the solutions of elasticity is only altered near the boundary where the equivalent boundary condition is prescribed, and for the place which is relatively far from the changed boundary, the solution will not be affected. If the boundary of the submodel is reasonably selected, and a fine mesh is used for the submodel, then high-accuracy result can be achieved.

In the analysis of three-dimensional cracks, the submodel method is composed of two steps: first a whole model is analyzed with a relatively coarse mesh, then a submodel cut from the whole model for the crack-front region is analyzed using direct generation commands (E and N commands), most (if not all) of the crack surface should be included in order to get high-accuracy results. By adjusting mid-side nodes to quarter-point, singular elements in wedge form are positioned directly along the crack front.

When analyzing crack problems, sometimes it is necessary to extend the submodel method to the partial crack submodel; that means only a part of the crack of interest is modeled in order to save modeling time. The partial crack submodel is also based on Saint-Venant's principle.

Singular elements generation using mesh200 element

Mesh200 element is "mesh-only" element, contributing nothing to solution. With the help of this element, we can generate three-dimensional crack elements easily. In order to generate three-dimensional crack elements, first the area mesh is generated with mesh200 element, using KSCON command to create two-dimensional singular area elements with 8 nodes at crack tip, then the volume mesh can be generated through VDRAG, VROTAT, VOFFST and VEXT commands etc. through a given coordinate system based on the area mesh. For complicated configurations, the model can be divided to many parts. These parts can be glued together or using bonded contact scheme. The one containing crack uses mesh200 elements to generate singular elements; the others can be meshed freely. It makes solving three dimensional crack problems easy.

Examples are given in the following paragraph to illustrate the usage of these methods.

Calibration of three-dimensional SIF using ANSYS

SIF for a penny-shaped crack in a finite-radius cylinder

Submodel method

This is a simple three-dimensional crack problem in finite domain, a penny-shaped crack in a finite-radius cylinder subjected to remote uniform tension. For this test problem, the crack radius a =0.5 (Figure 2), the radius of the cylinder b =1.0, the height of the cylinder h =2.8, the uniform tensile stress σ =1.0 is applied on both upper and lower surface, the elastic modulus E =20000.0, Poisson's ratio μ =0.3, all quantities are in compatible unit. The SIF should be identical along the whole circular crack front because of symmetry; its value given by the newest reference available is [2]:

$$\frac{K_I}{\sigma\sqrt{\pi a}} = 0.685$$

Where

$a =$

Radius of penny-shaped crack;

$\sigma =$

Tension stress;

$K_I / (\sigma\sqrt{\pi a}) =$

The dimensionless SIF.

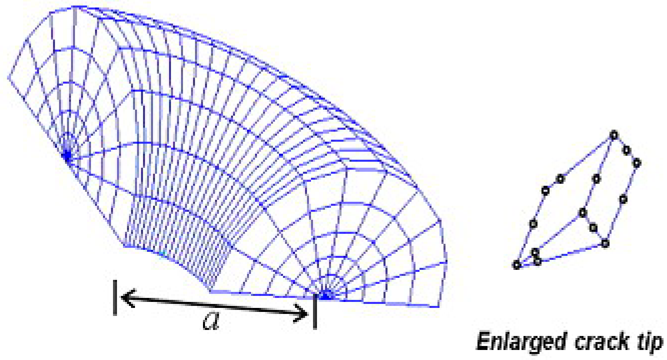


Figure 2. The submodel of the penny-shaped crack

Considering symmetry, only one-eighth of the cylinder needs to be modeled. First the normal solid tetrahedral element was used for establishing the whole model, the computation results was reserved for later use. Then a small region including the crack was separated from the whole model, the cut-boundary was given the prescribed displacement condition from previous computation for the whole model. This small region is the so-call submodel, the submodel covers the whole circular arc crack front, which spans the angle between 0° and 90°. The submodel was direct generated with another kind of solid hexahedral element, and the quarter-point element especially used to model the crack tip region. Using the submodel method we get the value of dimensionless stress intensity factor to be 0.681, its error as compared with Ref. [2] is -0.57%.

Partial crack submodel method

In order to test the partial crack submodel method which will be used in further research, only a part of the arc crack is modeled, which has the maximum angle of $\theta = 85.5^\circ$ as the cut boundary. It can be seen from Figure 3 that a part of the crack instead of the full crack (i.e. $\theta = 90^\circ$, where some exact symmetric conditions can be applied) is modeled, hence the name of the partial crack submodel method. When analyzing crack problems, sometimes is necessary to extend the submodel method to the partial crack submodel method. The computation results are shown in Figure 4, where it can be seen that the results of the partial crack submodel method are almost identical with those of the submodel method except the point very near the cut-boundary, this complies with the well-known Sanit-Wenant's principle.

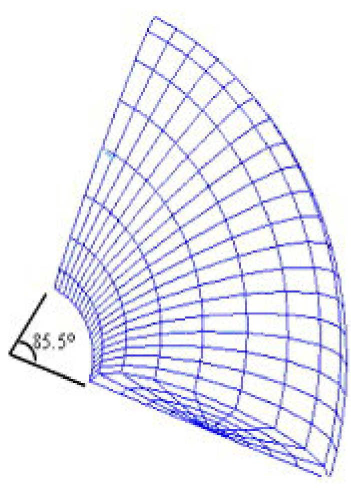


Figure 3. The partial crack submodel

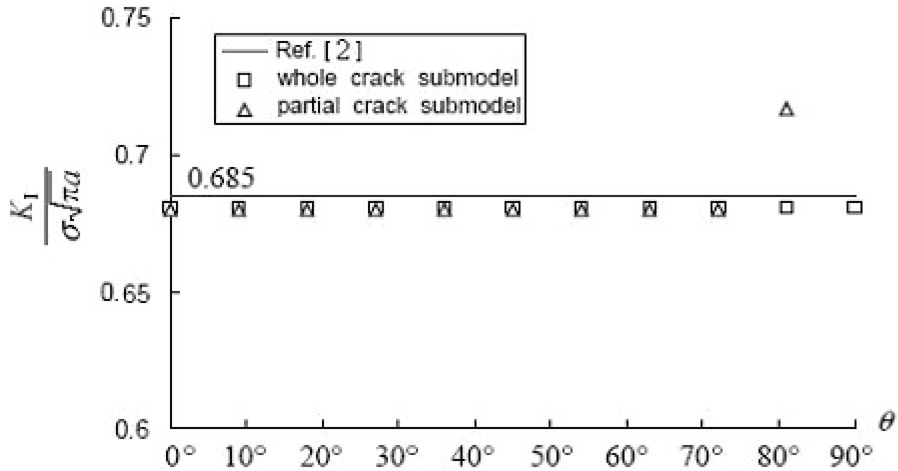


Figure 4. The comparison of the results between the whole crack submodel and partial crack submodel

Calibration of the minimum SIF for the CCNBD specimen

In 1995, the International Society for Rock Mechanics (ISRM) presented the "Suggested Method for determining Mode-I fracture toughness using cracked chevron notched Brazilian disc (CCNBD) specimens" [3]. The SIF computation of the CCNBD specimens is the most important part of the newly proposed Suggested Method; hence it deserves attention and research effort. It was pointed out that some background work of the SIF analysis for the CCNBD specimens were not appropriate in Ref. [4], the value given by ISRM [3], i.e. $Y_{\min} = 0.84$ are too small (where $Y_{\min}$ is the dimensionless SIF of standard CCNBD specimen). The CCNBD specimen with the concentrated diametric compressive load applied is shown in Figure 5, where R is the disc radius, B is the thickness, b the width of crack front, R_s the radius of the cutter, $\alpha_0(=a_0/R)$ the dimensionless initial crack length, $\alpha_1(=a_1/R)$ the dimensionless maximum cutting length, $\alpha(=a/R)$ the dimensionless crack length. Parameters for the standard CCNBD specimen given by Ref. [3] are: $\alpha_0 = 0.2637$, $\alpha_1 = 0.65$, $\alpha_B = 0.8$, $R_s/R = 0.6933$. For the convenience of modeling, the side circular arc notch, which has a finite notch width, is modeled as a crack, however in reality the notch is not a sharp crack, the singularity of stress at the notch root is less than that at the crack tip. In the experiment, the crack does not initiate from the root of the two side notches, the crack only initiates from the sharp conjunction point of the two side notches, and it advances forward with increasing crack front width (denoted by b in Figure 5) during testing. Based on this, our partial crack submodel will not include the two side notches; it only includes the center straight crack.

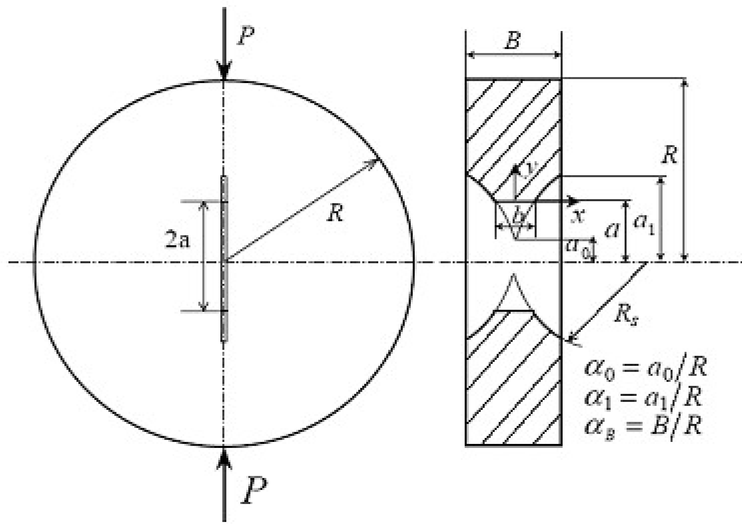


Figure 5. The CCNBD specimen

Because of the symmetry, only one-eighth of the CCNBD specimen is meshed in the whole model as shown in Figure 6. The whole model uses SOLID92 element from ANSYS library of element, while the submodel uses SOLID95, at crack front the SOLID 95 elements collapse into wedge-shaped with quarter-point elements instead (Figure 7).

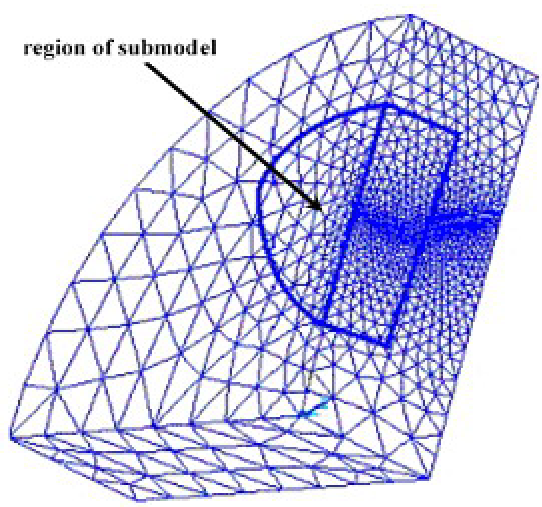


Figure 6. The mesh of the whole Model

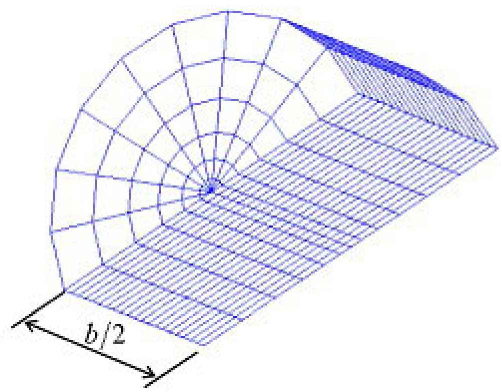


Figure 7. The mesh of the partial crack submodel

At different points along the crack front, the stress intensity factors are not the same, which is the characteristic of a three-dimensional specimen. In Figure 8(a), the distribution of the SIFs along the specimen for $\alpha = 0.49(\alpha = a/R)$ is given, where the SIF is normalized using $P/B\sqrt{D}$, the same quantity as SIF. The disturbance near the outmost point is relatively large , in order to avoid its interference, the average value of stress intensity factor is calculated for all those points whose value does not deviate from the center value for more than 7%.

$$Y = \frac{K_I}{P/B\sqrt{D}}$$

Where

$Y =$

The dimensionless stress intensity factor;

$K_I =$

The mode-I stress intensity factor;

$P =$

The concentrated diametric compressive load;

$B =$

The disc thickness;

$D =$

The diameter.

The formula for the determination of fracture toughness K_{IC} using CCNBD specimen is [3]:

$$K_{IC} = \frac{P_{\max}}{B\sqrt{D}} Y_{\min}$$

Where

$K_{IC} =$

The fracture toughness;

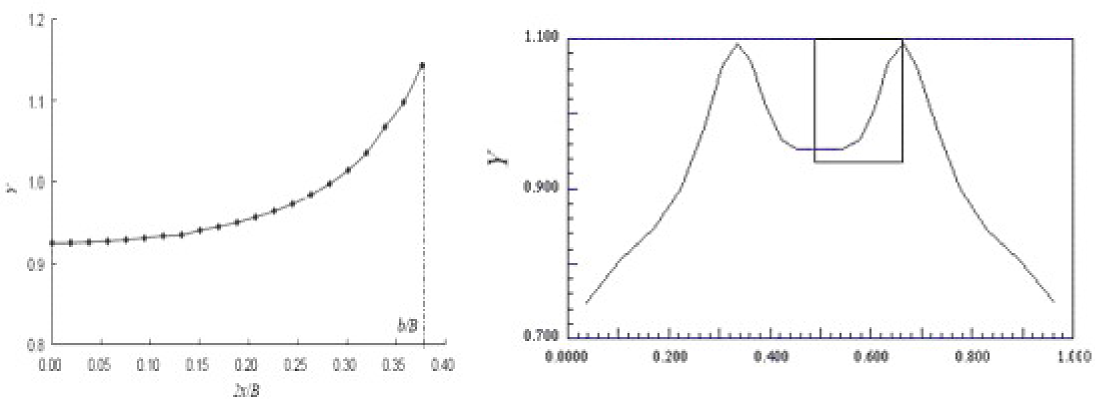
$P_{\max} =$

The ultimate load in the test;

$Y_{\min} =$

The minimum value of Y corresponding to the critical point in the test.

The general trend of the variation of SIF of the CCNBD specimen is similar to that of chevron notched specimens, that is descending-flat-ascending, and there is a minimum value of dimensionless SIF $Y_{\min}$. At this critical point, the load also reaches the maximum load $P_{\max}$, and the dimensionless crack length arrives at the critical value $\alpha_m (= a_m / R)$. For the standard CCNBD specimen proposed by ISRM, we obtained $\alpha_m = 0.49$ and $Y_{\min}(\alpha_m = 0.49) = 0.943$ using ANSYS submodel method, which is more accurate than the value given by ISRM (see Ref. [5] for detail). Figure 8(b) gives the SIF distribution along the whole crack front derived from FRANC3D (one famous three-dimensional crack analysis software). We can see that the distribution trend and the value are almost the same. Using FRANC3D, we get the value of the minimum SIF is 0.946. It indicates that the analysis using ANSYS partial crack submodel method is successful, the results are reliable.



(a) ANSYS (b) FRANC3D

Figure 8. The distribution of the dimensionless SIF of the CCNBD specimen with $\alpha = 0.49$

Dynamic SIF for penny-shaped crack in a finite cube

A penny-shaped crack in a finite cube ($2w_1 \times 2w_2 \times 2h$) under remote pulse impact $p(t) = \sigma_0 H(t)$ applied is shown in Figure 9, where a ($a/w_1 = 0.5$) is radius of the penny-shaped crack, $w_1/w_2 = 1$, $h/w_1 = 2$, Poisson's ratio $\mu = 0.2$ .Dynamic SIF is normalized with K_0 ,where $K_0 = 2\sigma_0\sqrt{a/\pi}$; time is normalized using formula $c_1 t / h$,where $c_1 (= \sqrt{(E(1-\mu))/(\rho(1+\mu)(1-2\mu))})$ is the longitudinal wave velocity.

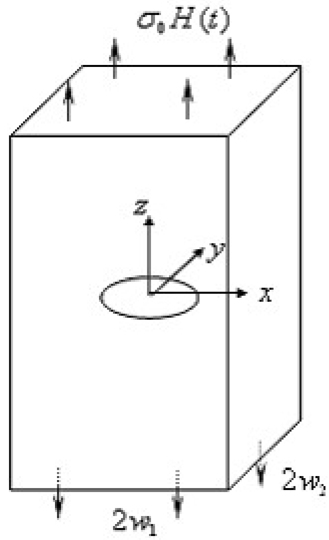


Figure 9. Penny-shaped crack in a finite cube

Considering the symmetry, one-eighth of the cube with the center penny-shaped crack is modeled. In order to generate crack mesh easily, the model is divided into two parts, the one is a one-eighth cylinder, and the other is a seven-side volume. In symmetry area, we use mesh200 element (KEYOPT(1)=7) and use KSCON command at crack tip to establish singular area mesh, then use VROTAT command rotate 90° to establish volume mesh containing three-dimensional crack elements. The seven-side volume are meshed freely also with SOLID95 element. The Mesh is shown in Figure 10, Figure 11 gives an enlarged view of three-dimensional crack elements, it can be seen that the element created are very fine.

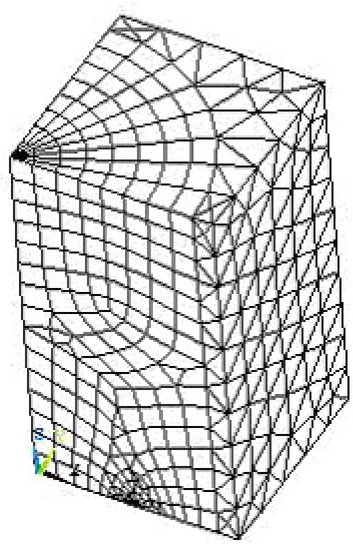


Figure 10. Mesh generation using mesh200

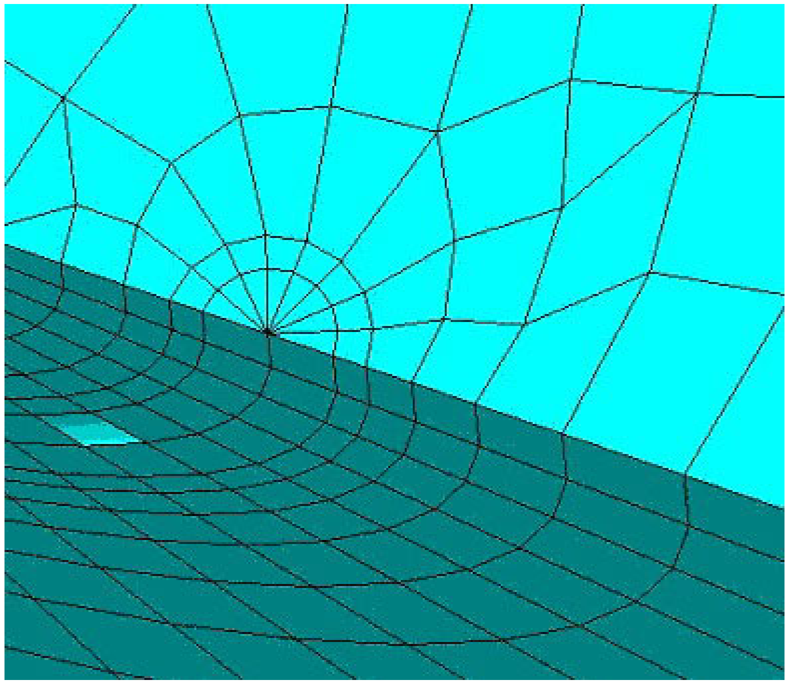


Figure 11. Enlarged crack-tip elements

The stress intensity factor should be identical along the whole circular crack front because of symmetry, the dynamic stress intensity factors at different time are shown in Figure 12, where it can be seen that the results using ANSYS are almost identical with those of the reference values [6, 7], the maximum error is 1.2% in calculating time. It is clear that we can generate well formed three-dimensional crack elements and get high-precision results in SIF computation with the help of mesh200 elements.

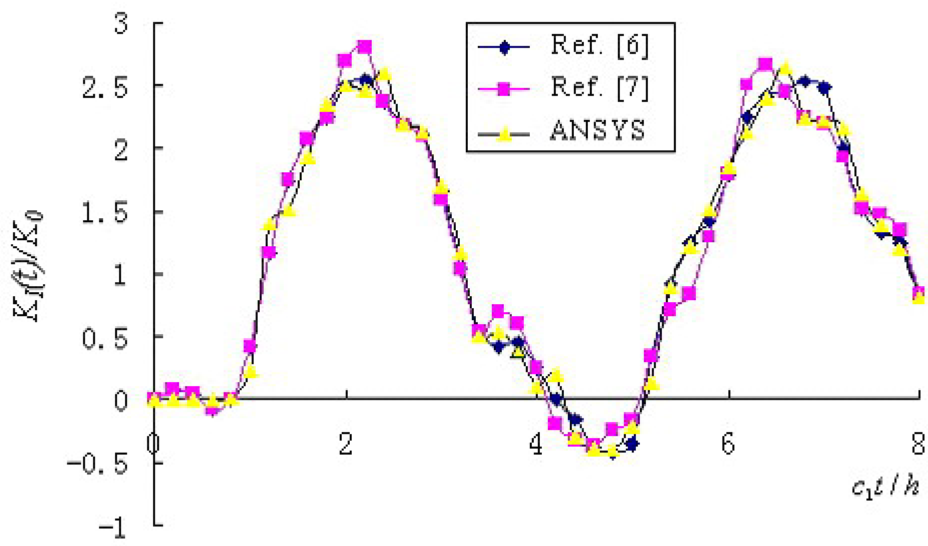


Figure 12. Dynamic stress intensity factor of the penny-shaped crack in a finite cube

Conclusion

In this effort, two methods are presented to compute the three-dimensional static and dynamic stress intensity factors, the examples given prove the correctness of the methods, and these methods make the computations easy and high efficient. The results are of high precision and reliability compared with other literature. These two methods are easy to handle and extend the ability in computing three-dimensional SIFs.

References

1. ANSYS Reference Manual [M]. ANASY company, 1999.
2. Leng, A. Y. T., Tsang, K. L., Two-level finite elements for a finite body with penny shaped crack [J]. Int. J. Fract., 1999, 100:9-L14
3. ISRM Testing Commission, (co-ordinator: R. J. Fowell), Suggested method for determining mode I fracture toughness using cracked chevron notched Brazilian disc (CCNBD) specimens [J]. Int. J. Rock Mech. Min. Sci. Geomech. Abstr., 1995, 32:57-64.
4. Wang Q Z., Stress intensity factors of ISRM suggested CCNBD specimen used for mode-I fracture toughness determination [J].Int. J. Rock Mech. Min. Sci. 1998, 35: 977~982
5. Q. Z. Wang, X. M. Jia, et al. More accurate stress intensity factor derived by finite element analysis for the ISRM suggested rock fracture toughness specimen-CCNBD [J]. Int. J. Roc Mech. Sci. 2003, 40: 233~241.
6. Zhang Y Y. Shi W, Transient analysis of three-dimensional crack problems by the Laplace transforms boundary element method [J]. Engage Fracture Mech,1994,47(5):715-722
7. P. H. Went. Dynamic Fracture Mechanics: Displacement Discontinuity Method [M]. Southampton , Boston: Computational Mechanics Publications ,1996