Memoirs of the School of Engineering,Okayama University,Vol.21,No.2,February 1987

Some Remarks on Finite Element Mesh Modeling of Crack-Tip Area

Takeo TANIGUCHI*, Kenji SANADA*, Hajime MATSUMOTO* and Kiyoaki MORIWAKI*

(Received January 16, 1987)

SYNOPSIS

The aim of this paper is to present some remarks on the arrangement of finte element mesh modeling of the area adjacent to the crack-tip which locates in two-dimensional area. Since the stress distribution near crack-tip is singular, the arrangement of mesh pattern and the selection of mesh type in the crack area govern the accuracy of the solution. This paper gives some informations on the arrangement of finite elements in the area which are obtained through numerous number of numerical experiments. And the effectivity of Zooming Technique for stress analysis is clarified through the experiments.

1. INTRODUCTION

Recently we find some technical papers which report the appearance of cracks in civil engineering structures. But, in USA many reports and books have been already published, and many investigators are now studying the phenomena.

There are two aspects to the problem of CRACK; One of them is how to prevent the occurance of crack, and another is how to prevent the propagation of its growth or how to estimate the residual life of the structure.

For cracks found after the construction the latter problem is important, and it includes following questions;

* Department of Civil Engineering

1. How has the crack propagated?
2. How long residual life is expected for the structure?
3. How can engineer repair the wounded places ?

One of the best ways to answer to these questions is to develop the simulation method of crack propagation, and as the tool Finite Element Method is recognized to be effective and convienent because of its flexibility of ,for example, the setting of area for the analysis, its boundary condition and so on. But, at its application there exist some unresolved problems which are originally caused to the method and the problem.

Finite element method can give only approximate solution, and the solution is largely governed by the characteristics of finite element being used and the mesh arrangement. On the other hand, the actual stress distribution in the region adjacent to the crack-tip is singular, and the behaviour of crack is determined by stress, strain, and displacement at the small area locating the crack-tip. This indicates that the finite element model in the area governs the behaviour of crack at the numerical simulation of crack propagation.

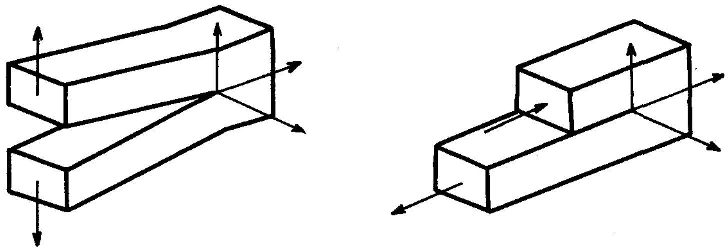
In this paper the authors aim to propose an appropriate finite element modeling procedure of two-dimensional area with a crack for conventional finite element method. Through a number of numerical experiments the propriety of the proposed mesh arrangement is examined, and further informations of the application of finite element method to the stress analysis are also presented in this paper.

2. STRESS INTENSITY FACTOR

The stress distribution near crack-tip located in two-dimensional area is expressed as a function of the location by using local polar coordinate system fixed at the crack-tip and it has the singularity of $r^{-\frac{1}{2}}$,where r is the distance from the crack-tip, as shown in eq.1. 2)

$$\sigma = \frac{K(i)}{\sqrt{2\pi r}} f(\theta) \tag{1}$$

K(i) in eq.1 is called the stress intensity factor, and in the in-plane problem there exist two kind of factors, K(I) and K(II), for different displacement modes shown in Fig.1. $f(\theta)$ in the equation is the function of the angle between the axis along the crack-direction and the position.



Mode I　　　　Mode II

Fig. 1 Fundamental Modes

The displacement of the position is also given as following by using two stress intensity factors;

$$u = \frac{1}{2G}\sqrt{\frac{r}{2\pi}}\ \{\ K(I)g' + K(II)g''\ \} \tag{2}$$

, where G is shear mudulus, and g' and g'' are the function of the angle and Poisson's ratio. Setting the angle in above equation to be 180 degree we obtain the stress intensity factors by using the displacements v and u along y and x axes, respectively, which located at the crack-tip;

$$\begin{aligned} K(I) &= \sqrt{2\pi/L}\ \frac{G}{\kappa+1}\ (v'_C - v'_E) \\ K(II) &= \sqrt{2\pi/L}\ \frac{G}{\kappa+1}\ (u'_C - u'_E) \end{aligned} \tag{3}$$

, where $\kappa = 3-4\nu$ for plane strain analysis and $\kappa = (3-4\nu)/(1+\nu)$ for plane stress, and L is the length of crack-tip element.(See Fig.2)

Another expression of the factors can also be given by using the displacements of more nodes adjacent to the crack-tip. 1)

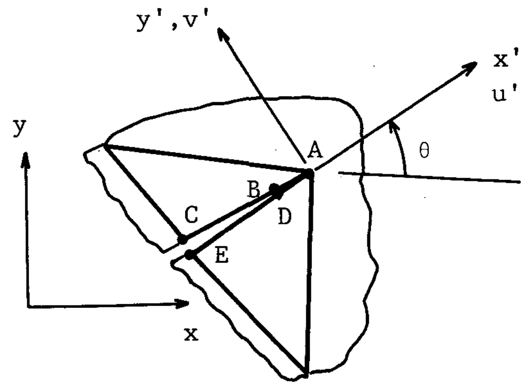


Fig. 2 Crack-tip Elements

$$\begin{aligned} K(I) &= \sqrt{2\pi/L}\ \frac{G}{\kappa+1}\ \{\ 4(v'_B - v'_D) + v'_E - v'_C\ \} \\ K(II) &= \sqrt{2\pi/r}\ \frac{G}{\kappa+1}\ \{\ 4(u'_B - u'_D) + u'_E - u'_C\ \} \end{aligned} \tag{4}$$

where L is the distance between A and C in Fig.2. Note that above expression is valid only for the singular isoparametric triangular element with 6 nodes. Details of eq.4 should be refered to Fig.2.

Following integration

$$J = \int_{\Gamma} \left(U_0 dy - T \frac{\partial u}{\partial x} ds \right) \tag{5}$$

is J-Integral and it is also one of important mechanical factors in fracture mechanics. In this equation U is the strain energy density, and the line integral must be anticlockwisely integrated from a point on an edge to another point on opposit edge of the crack so that the integration path encloses the crack-tip. Since J value obtained by eq.5 is related to the energy release rate which corresponds to the change of strain energy due to the crack propagation, J is also connected to the stress intensity factor as shown in eq.6. But, note that the stress intensity factor obtained from J value cann't distinguish the difference of two modes and it can estimate only K for a single mode, K(I) or K(II), or it can be recognized as K obtained by using eq.7.

$$J = K^2/E \tag{6}$$

$$K = [\ (K(I))^2 + K(II)^2\ ]^{1/2} \tag{7}$$

The mechanical behaviour , for example, the growth of the crack length and its direction, adjacent to the crack-tip is described by using the stress intensity factors, and this indicates the accuracy of K governs them.

There exist three methods to estimate the stress intensity factors as mentioned above, i.e. eq's 1, 3 or 4, and 5. In the first method called Stress Method K's are directly derived from the stresses of a point, the second method called Displacement Method obtains them by using displacements of nodes locating near crack-tip, and the last method using J-Integral leads to K by stress distribution surrounding the crack-tip. Then, there arise problem how we can obtain the stress distribution and the displacements near the crack-tip as accurate as to use for above methods. Let's consider this problem from the point of view of using the result of the finte element method.

Firstly we must note that the result by Finite Element Method is only an approximate one and it is largely governed by mesh arrangement and the characteristics of the element. We assume to apply the singular isoparametric elements at , at least, the area near the crack-tip in order to express the singularity of the stress distribution accurately. Even if the element can express the stress singularity sufficiently, there remains the problem how to set the meshes in the area. This problem includes the problem of modeling of the boundary conditions, i.e. forces and displacements on the boundary.

Successively we must note that the cost of computation must be taken into consideration at the application of Finite Element Method to actual engineering problem. The term "cost" includes the computation-time and also the necessary memory size. The nature of the method necessarily requires finer mesh model for better result, but there exists apparant limit for them as far as a computer is used as a tool. Then, the problem is how many elements are necessary to get sufficiently accurate solution which can satisfy the user.

The main purpose of this investigation is to answer above two questions, and the method mainly due to numerical experiments which are summarised as followings;

Problem	:	Center cracked plate tension specimen (CCT) Single edge cracked three point bending specimen Single edge cracked tension specimen (SECT) Center inclined cracked plate tension specimen Single edge inclined cracked tension specimen
Solver	:	Finite Element Method (Displacement method)
Element	:	Near crack-tip by Singular triangular isoparametric element with 6 nodes, and residual area by triangular / quadrilateral isoparametric elements
Mode	:	Mode I and II

3. EVALUATION METHOD OF STRESS INTENSITY FACTOR

The main purpose of this section is to survey the modeling of the area adjacent to the crack-tip for the finite element analysis. But, it is expected the evaluated values are influenced not only by the finite element models but also by the evaluation methods. Thus, before treating our main theme of the finite element modeling of crack area we examine the difference of K's according to the different tools.

Comparison of Three Estimation Methods

The aim of this section is to survey the effectivity of above three estimation methods of the stress intensity factor, i.e. the stress method, the displacement method and J-integral method.

A simple structure shown in Fig.3 is used for this purpose. The figure shows an infinite strip with the width 2W subjected to the tensile force along one axis, and we assume the crack length 2a. This case shows the crack-opening mode (Mode I), and K(I) is theoretically obtained as

$$K(I) = \sigma_0 \sqrt{\pi a}\ F(a/W) \tag{8}$$

, where σ_0 is the average tensile stress along the loading axis, and the function F(a/w) is a function of the ratio of a/W and is explicitly expressed for the ratio of the span and the height of specimen. (Refer Appendix of 3))

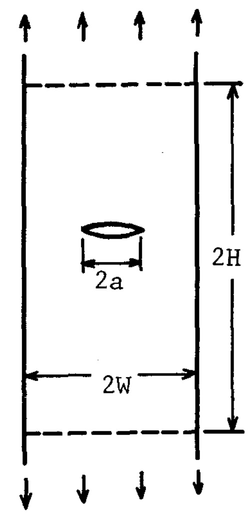


Fig. 3 CCT Model

K(I) for the stress method is evaluated at an element which locates near to the crack-tip and also satisfies the condition of angle = 0 degree. For the displacement method the value is obtained by using displacement of some nodes locating near the crack-tip and satisfying = 180 degree. In this case eq's 3 and 4 are applied for its evaluation. In case of J-Integral several pathes for the integration are considered in order to check the condition of path-independence.

All of these results are presented in Table 1, and the conclusion is summarized as following; The displacement and J-integral methods can give good coincidence with the theoretical value, but the difference of results between the stress method and the theory is relatively large. The reason is that since the applied finite element method is the displacement method, with the same number of nodal points it can give better result for displacement than for stress. This suggests that if more nodes are set in its model, better result is obtained for the stress method, too.

Table 1 Accuracy of Stress Intensity Factor

Method	Accuracy (%)
Stress Method ; Eq.1	92.4
Displ. Method ; Eq.3	102.4
Displ. Method ; Eq.4	97.1
J-Integral ; Eq.6	110.5

Table 2 Accuracy of Stress Intensity Factors

Model	Displ. Eq.3(%) K(I)	K(II)	Displ. Eq.4(%) K(I)	K(II)	J-Integral K
SECT	100.7	-	98.6	-	104.4
TPBS	103.2	-	96.6	-	103.4
CCT	96.4	-	100.7	-	103.8
ICCS (30)	96.3	95.3	99.2	99.9	96.3
ICCS (45)	95.9	94.6	99.3	99.8	98.8
ICCS (60)	96.9	95.9	100.9	101.2	101.9

Notes: Values ; K by FEM/K by other method
TPBS ; Single edge cracked three point bending specimen
ICCS ; Center inclined cracked plate tension specimen
(30) ; Inclined angle=30 degree

Now, we examine the difference of the stress intensity factors obtained by using two kind of displacement methods, i.e. eq's 3 and 4 in previous section. As indicated in previous section the difference of these two equations is the number of nodes whose displacements are introduced in the evaluation of K. The results of numerical experiments are summarized in Table 2. Models treated in the experiments are SECT, Three-points Bending Problem, CCT and Rectangular Plate with Inclined Central Crack models. K(I) and K(II) in the table present the accuracy of the stress intensity factors of presently computed values to values which are obtained theoretically, experimentally, or by using other methods. The values in ( ) shows the inclined angle of the crack from y-axis.

The results show that eq.4 can lead to better evaluations of K comparing to eq.3. Since the displacement method evaluates K by using the displacements of nodes in elements adjacent to the crack-tip, the accuracy of K is wholly governed by the deformed configuration of the crack. In eq.4 we use all nodal displacements of elements adjacent to the crack-tip, and it results in above conclusion. Henceforce we use eq.4 for the evaluation of the stress intensity factors.

The last column of Table 2 shows the stress intensity factors which are evaluated by using J-integral method. The first three cases are strictly for experiments of Mode I deformation, and, therefore, K by J-integral coincides with K(I). But the last case of rectangular plate with inclined central crack behaves as mixed mode deformation of Mode I and Mode II. Then, the evaluation of K for these cases with different angles is done by eq.7. Comparison of the results of K's by the displacement and J-integral methods can lead to the conclusion that J-integral method is also as effective as the displacement method. But, note that the method cann't distinguish the difference between two modes. Thus, in successive numerical experiments we use J-integral method as a tool for cross-checking of K values by the displacement method.

Finite Element Mesh Model and Stress Intensity Factor

Finite Element Method can treat only a definite area, and there arise following fundamental problems when it is applied to actual crack problem or when its solution is compared with theoretical solution. That is, if the infinity for the boundary of two-dimensional area is assumed for the theoretical method, the problem cann't be directy treated by FEM and some boundary condition must be assumed for the area.

In actual problem the length of crack is very small comparing with the dimension of the area, and the modeling for the analysis is quite difficult. In this case too we have to assume some boundary appropriately.

Summarizing above considerations there exist following problems which must be treated in this section;

(1). Modeling of Relatively Small-scale Area

(2). Modeling of Large-scale Two-dimensional Area

(3). Modeling of Infinite Two-dimensional Area

These three problems of modeling are successively treated through numerical experiments.

(1). Modeling of Relatively Small-scale Area .

The modeling procedure of the case where the dimension of the area is relatively small comparing with the length of the crack is treated here. It is wellknown that the stress obtained by using FEM is largely influenced by the characteristics of element and also the arrangement of elements. Since the main purpose of present study is the stress analysis near crack-tip, we use the isoparametric triangular element with 6 points and isoparametric quadrilateral element with 8 points for the modeling of the area not adjacent to the crack-tip and singular isoparametric triangular element with 6 points as the crack-tip elements. Then, our aim is to survey the arrangement of these elements, and there exist two kind of problems; the first is how small

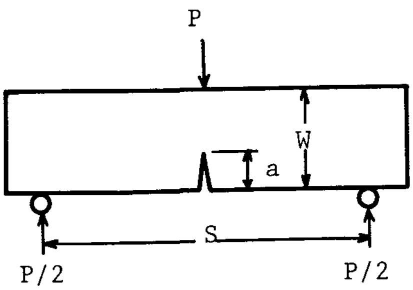


Fig. 4 Three Point Bending Specimen

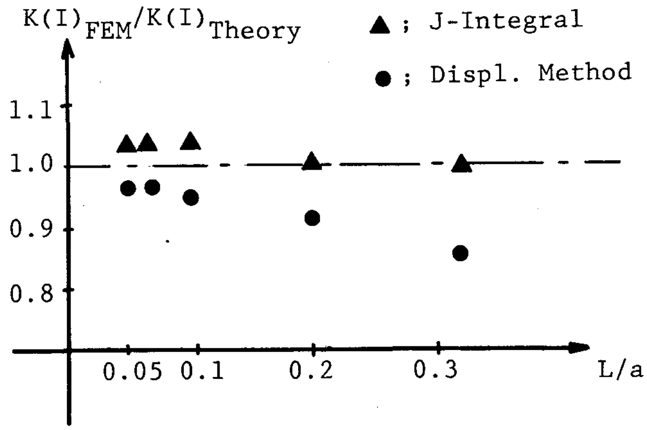


Fig.5 Influence of Element Size

elements are required for modeling the area adjacent to the crack-tip and the second problem is how to arrange them at the region.

Single edge cracked three point bending problem is treated as an example for this purpose (see Fig.4). We assume a/W =0.5 and examine the accuracy of computed stress intensity factor due to the ratio of the element length and the half length of the crack (L/a). The result is illustrated in Fig. 5. The result shows that the accuracy depends on the ratio, and in order to get accurate solution the ratio L/a should be less than 0.1. Henceforce, we use this value for our numerical experiments.

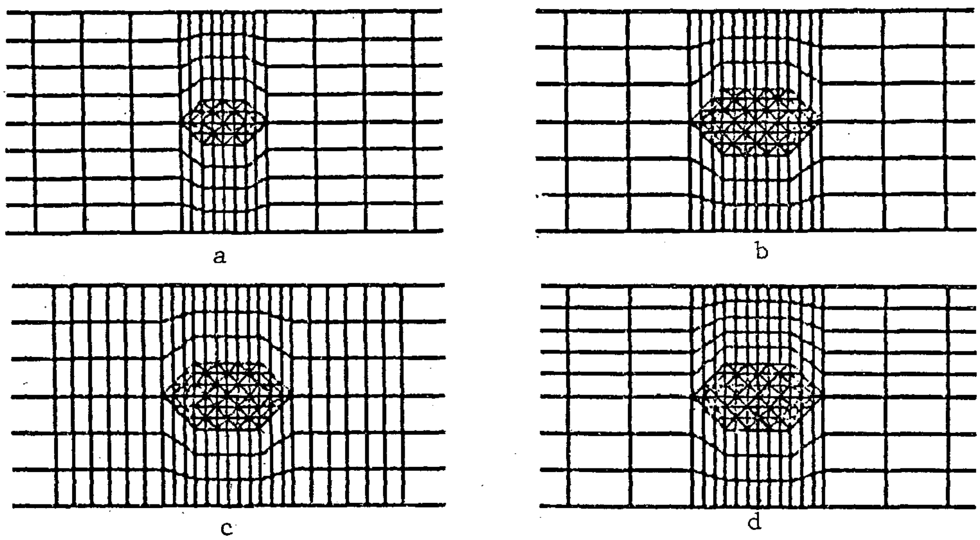
The same model is used for the examination of the accuracy of solution due to the different arrangement of isoparametric elements. It is recommended that at the crack-tip there should exist more than six elements, and we place 8 singular isoparametric triangular elements at the crack-tip. (See Ref.1) Since the homogeniety of elements is required at the small area, the problem to be solved is how many elements are required to model the area. Different number of element layers are arranged at the area, and also different mesh sizes are prepared for the modeling the residual area adjacent to the small area. The results are summarized in Table 3. According the table we can conclude that 1). at least three layers of elements should be placed for modeling the small area of the crack-tip, and 2). the residual area may be roughly discretized if three layers of small elements are prepared so that they surround the crack-tip.

Summarizing the results obtained in this section we can suggest that 1). the ratio of the length of element and the half of the crack-length should be less than 0.1, 2). at least three layers of small

Table 3 Influence of Finest Element Layers to Accuracy

Numbers of Layers	K by Displ. Method	K by J-Int.	Fig. No.
2	62.7	98.2	a
3	94.9	103.5	b
3	94.9	103.5	c
3	94.9	103.5	d

(%)



elements must be placed as to surround the crack-tip, and 3). the residual area may roughly discretized if the second item is satisfied.

(2).Modeling of Large-scale Two-dimensional Area

Fig. 6 shows the relation between the mesh size and the accuracy of the computed result, and this suggests that finer mesh size gives better result. On the other hand, actual crack-length appearing in a structure is generally very small comparing with the size of the structure, and FEM user encounters the difficulty of modeling of the structure with the crack. If the user aims to model a structure so that the mesh satisfies the conditions mentioned above, the number of elements included in the model necessarily increases and the arrangement of homogeneous mesh pattern becomes difficult. In order to prevent the increasing of the number of nodes and elements a technique called "Zooming" is effective, and in this section we survey the efficiency of this technique.

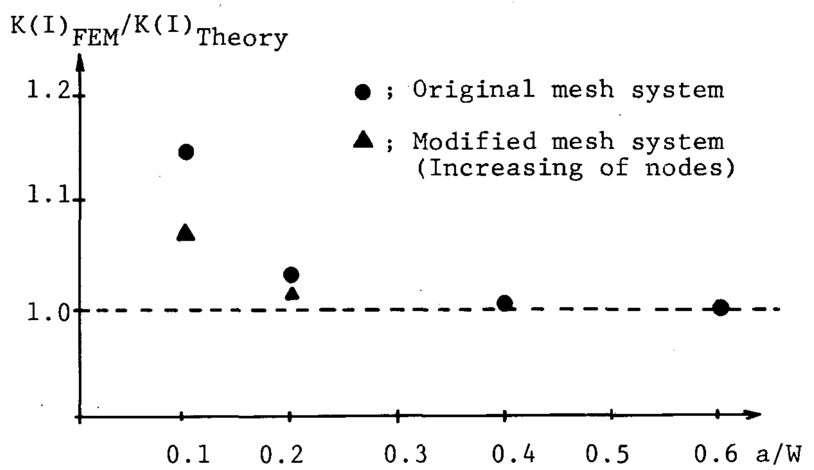


Fig.6 Improvement of Accuracy by Increasing of Nodes

Zooming method is explained as following; Original structure is at first analyzed as a whole, and a part of the solution (nodal displacement vector) which locate as to surround the area including the crack-tip is introduced as the boundary condition for the reanalysis of the smaller area. This procedure is repeated till the ratio of the dimension of the area and the crack length satisfies the condition obtained in above section.

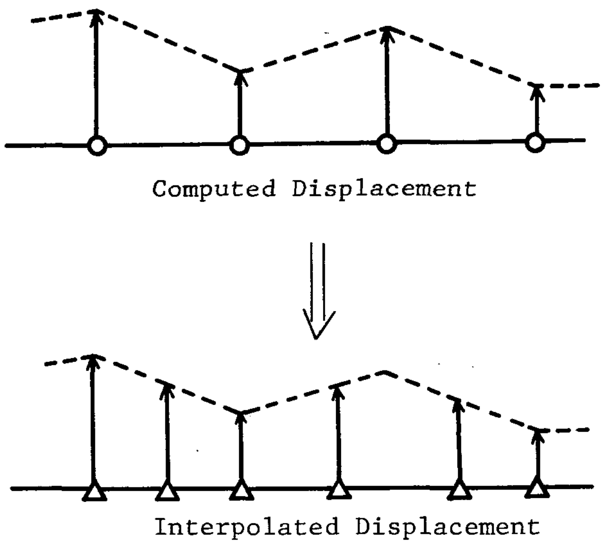


Fig.7 Zooming Method for Boundary

At the application of this method the appropriate interpolation method of the displacement vector in previous step to successive boundary condition is required, and there exist a number of methods for this purpose. In this study we use one of the simplest methods, i.e. the linear interpolation method (see Fig. 7).

In Table 4 and 5 the results of numerical experiments are summarized. The results show that this technique is effective not only for improving the accuracy of the stress intensity factor but also for saving the execution-time and necessary memory.

The introduction of Zooming Technique can remove the difficulty of

Table 4 Effectivity of Zooming Technique

		K by Displ.(%)	K by J-Int.(%)	Nodes	CPU(sec)
With	Before	113.7	100.5	246	8.3
Zooming	After	96.5	104.5	396	28.0
Without	Model 1	97.0	99.5	644	55.2
Zooming	Model 2	98.8	102.1	984	104.2

Table 5 Effectivity of Zooming Technique

		K by Displ.(%)	K by J-Int.(%)	Nodes	CPU(sec)
With	Before	115.5	119.2	240	8.4
Zooming	After Model1	95.9	103.9	323	26.4
	After Model2	95.8	103.8	397	32.1
Without	Model 1	88.9	87.0	644	54.3
Zooming	Model 2	93.3	-	984	102.2

generating homogeneous mesh pattern, and it gives good influence for the evaluation stress intensity factors.

(3). Modeling of Infinite Two-dimensional Area

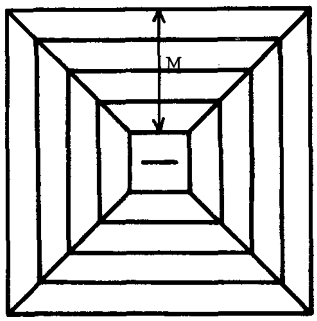
Some of theoretical solutions in fracture mechanics can be obtained by the assumption of the infinity to its boundary condition. In actual cases the dimension of the area is large enough to be thought as infinite comparing with the scale of the dimension of the crack-length. At the application of FEM to these cases we have to give an appropriate boundary, and its determination is treated here.

If a crack exists in an infinite plane, the area which should be treated is the finite region whose boundary condition gives no influence to the value of K. Table 6 shows the result of simple experiments for this purpose. In this experiment we assume the ratio

Table 6 Influence of a/W and Outer Meshes to K

a/W \ M	2	4	6
0.4	118.6	118.6	118.6
0.2	107.2	107.3	107.3
0.1	103.3	104.3	104.4
0.45	99.5	102.4	103.2

(%)



of the length of element and the half length of crack (L/a)=0.1. The result can lead to the conclusion that the width of area (w) may be 10 times of the half length of the crack, i.e. a. The number of layers of elements for modeling this outer area should be more than 4, and this suggests that at least 4 layers of elements are necessary to remove the influence of the boundary condition.

Semi-infinite case is also treated in this study. By introducing the result of infinite case the region is bounded at the distance of 10 times of the crack length, and L/a is set to be 0.1 in this case, too. The results are compared to the ones by the colocation method, and they show good coincidence. (See Fig.II.14 of Ref.3)

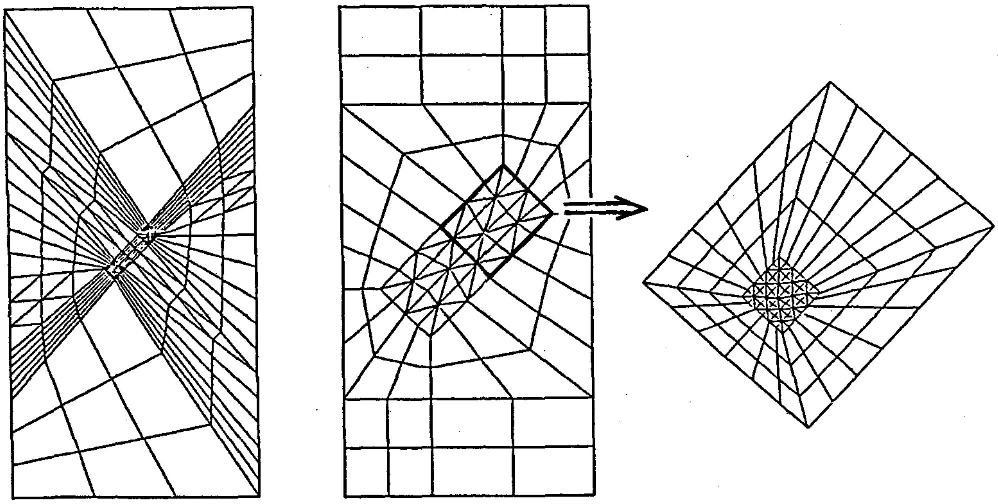
An infinite strip with an center crack is also treated and the result is compared with the result in Table I.I of Ref.3. Same condition for the finite element model is applied in this case, too, and the results show good coincidence with the table.

Above three numerical experiments indicate us that the infinity of the area may be replaced by the area with edge length of 10 times of the crack half-length.

The result of Fig.6 shows that the accuracy of the stress intensity factors can be improved in accordance with the increase of nodal points set in the modeling of the structure. But, the restriction of CPU-time and necessary memory size forces the FEM user to save them, and this causes the origin of the compexity of stress analysis in the crack problem. This compexity becomes more serious if mixed mode fracture is treated, because the appropriate rearrangement of meshes generally becomes more difficult comparing with the case of single mode fracture. Fig.8 shows a simple plate with a crack which behaves as mixed mode fracture. If the crack locates along the direction of the width, then it behaves as Mode I and the arrangement of homogeneous meshes for the structure is easy, but the case in Fig.8 necessarily requires more nodes for the purpose. If its modeling is done by using the same number of nodes of the model for single mode ,then the homogeniety cann't be hold.

Now, we try to apply "zooming technique" for this case. At the analysis of the original structure we need not to take care of the accuracy of the stress distribution at the crack-tip but only the accuracy of the displacement of nodes which enclose the region of successive analysis, and this makes ease the modeling of whole area. At this stage we set an appropriate region for the successive stress analysis, and the solution vector of this surrounding boundary is introduced in successive analysis as boundary condition. Since the area which is treated after the zooming is simple enough, homogeneous mesh system is easily obtained by less number of nodes comparing with the direct mesh generation of the original structure.

One example presented in Fig.8 is used for our experiment, and the results are summarized in Table 7. The results show the effectivity of Zooming Technique for stress analysis of mixed mode fracture. And, this result is easily extended to the case of stress analysis of structure with complex boundary configuration.



(a) Without Zooming Before Zooming After Zooming (b) With Zooming

Fig. 8 Mesh Systems with and without Zooming

Table 7 Effectivity of Zooming Technique

	K(I) (%)	K(II) (%)	Nodes	CPU(sec)
Case 1				
Before Zooming	110.2	109.9	296	11.0
After Zooming	95.9	95.4	396	28.5
Case 2				
Before Zooming	104.6	105.6	602	37.4
After Zooming	97.3	97.5	396	28.5
Without Zoom	97.4	96.9	864	99.1

Note : K is evaluated by Displacement Method.

4. CONCLUDING REMARKS

In this paper the authors investigated the mesh modeling method for finite element analysis which can offer accurate stress intensity factor. The models treated in this paper are very common cases like CCT, SECT, Single Edge Cracked Three Point Bending Specimen, Center Inclined Cracked Plate Tension Specimen, Single Edge Inclined Cracked Tension Specimen, and the computed values are compared with the theoretical, experimental, and also other numerical results.

According to the results the finite element mesh systems which can give appropriate values are summarized as followings;

1). Mesh system must be homogeneous.

2). The mesh size adjacent to the crack-tip must be finer than 1/10 of the crack length.

3). These finest meshes must be arranged at least three layers so as to surround the crack-tip.

It is difficult to make satisfy these conditions for any case of crack analyses. The numerical experiments in this investigation clarified that the zooming technique is effective to remove the difficulty, and its introduction to the stress analysis can make FEM user easy to generate satisfatory mesh model. Furthermore, the technique can save not only CPU-time but also the necessary memory size. That is, Zooming Method is effective for the improvement of the solution and cost saving.

ACKNOWLEDGEMENT

The authors thank Mr.Tetsuya Takasaki for his help in preparing Mesh Generator for Crack Analysis used in this study.

REFERENCES

1). A.R. Ingraffea,"Numerical modeling of fracture propagation",Rock Fracture Mechanics (ed. H.P. Rossmaith), Springer-Verlag (1983), 151-208

2). H. Murakami & M. Oonami,"Introduction to fracture mechanics", Ohmu-sha (1979) In Japanese

3). T. Kunio et al,"Experimental methods of fracture mechanics", Asakura-shoten (1984) In Japanese