Memoirs of the Faculty of Engineering, Okayama University, Vol.24, No. 1, November 1989

Numerical Simulation of Multicrack Propagation Behaviour in Steel Structure

Takeo Taniguchi*, Tsuyoshi Suetsugu** and Masaki Iwasaki***

(Received October 16 , 1989)

SYNOPSIS

This paper describes the numerical simulation method of the multi-crack propagation behaviour which appear in a part of civil engineering structures with complex geometrical configuration like steel bridges. Proposed method can treat the interaction of several cracks which locate in a short distance each other, and the process of their growth can be grasped. The method is based on the finite element method, and the linear fracture mechanics is assumed. Proposed method includes following tools for the simulation of the crack propagation behaviour: Automatic Mesh Generators for 3-D, 2-D structural analysis, and 2-D crack propagation analysis, Multi-level Structural Analysis Technique, Estimation Method of the crack growth and the angle of cracks and the modelling method of traffic loadings. The validity of the method is investigated by comparing the result to the experimental one.

1. INTRODUCTION

The crack propagation behaviour in structures due to the repeated loadings is generally investigated by the structural experiment, but it not only takes long time but also costs too much. Moreover, the adjustment of the experimental conditions, for ex-

* Department of Engineering Science ** Department of Civil Engineering
*** Yokogawa Bridge Works, LTD.

ample the boundary condition and the dimensions of the specimen, is very difficult.

On the other hand the numerical simulation method is gradually recognized in accordance with the developments of the digital computers and the computational method like the finite element method. The main reason is the easiness of the setting the complicated boundary conditions and the physical conditions. The subsidiary reason is its cost and the quickness. As far as the numerical simulation method is used for our investigation, the difficulties which occures at the structural experiments disappear.

Theoretical and experimental investigations on the crack behaviour in 2-dimensional structures have been done, and at present the behaviour in 3-dimensional space is under the investigation. The numerical investigation of their behaviour also have been done, and some papers have also published.

But, their simulations are limited in the crack propagation behaviour in a structural member and not in a member included in a complicated structure.

In this paper we propose a general numerical simulation method for investigating single and multiple crack(s) propagation phenomena in 2-dimensional area. It can be applied not only to cracks in a structural member but also to a structural member in a complicated structure like steel bridges. This paper shows the details of the numerical simulation technique, and several test problems are solved and their solutions are compared with the the experimental ones.

2. NUMERICAL SIMULATION OF CRACK PROPAGATION BEHAVIOUR

We assume several cracks initially locating in neighbourhood in an arbitrary 2-dim. structural member which is a portion of a complex structure, and our aim is to simulate their propagation behaviour due to applied loadings. The load to be considered is the repeated ones with random intensity like traffic load.

The actual crack propagation behaviour is governed by the stress distribution near the crack tip, and its circumstance changes at every instance of the crack propagation. For example, following items give serious influence to the stress distribution; the location of the crack-tip, the geometrical condition near the crack-tip, the geometry of the structural member where the crack locates, the structure containing the structural member, and the boundary conditions of the loading and the constraint of the structure. Moreover, if several cracks locate very closely each other, the stress distributions near crack-tip areas give influence each other and, as a result, we can guess that their propagation phenomenon becomes very complex.

Theoretical investigations on the physical condition near crack(s) have been done for a number of spesific boundary conditions 1), but we encounter difficulties at the application of these results to actual crack propagation problems. Especially, these difficulties come from the difference of the boundary conditions, the geometrical conditions of the area where the crack(s) locates, and the geometry of crack(s).

Now, we try to solve the crack(s) propagation behaviour numerically. As indicated already, the most important and serious problem of the crack(s) propagation analysis is the treatments of the boundary conditions, and the geometries of cracks and their structural member where cracks locate. The best tool for these problems is the finite element method, and we introduce it to our system. In next section we consider on the details of the necessary techniques which are required for the simulation method of the crack(s) propagation behaviour.

3. TOOLS FOR NUMERICAL SIMULATION OF THE CRACK(S) PROPAGATION BEHAVIOUR

3.1 Necessary Tools for The Numerical Simulation based on FEM

(1) The geometrical largeness and the complexity of the structure treated in our investigation requires numerous number of meshes for its finite element modelling. At the same time, in order to express the singular stress distribution near the crack-tip area, very fine meshes must be set at the area. Then, it is obvious that these two requirements for the modelling cann't be satisfied at the same time, and, therefore, some effective tool of the structural analysis is required.

(2) The behaviour of any crack can be expressed by using the stress

intensity factors, and at present there are several methods to estiamte these factors. Since these factors govern the behaviour of the crack propagation, they must be estimated accurately.

(3) It is wellknown that whether any crack propagates is determined by the stress condition at the crack-tip. Then, a criterion for this judgement is required.

(4) Considering the characteristics of the digital computer, the numerical simulation is based on the repetition of the stepwise analyses of the behaviour of cracks. Then, at each analysis step, the direction and the increment of the growing crack must be calculated for the loading condition.

(5) The load to be considered in this investigation is the trafic load which varies randomly. Since it is impossible to introduce this characteristics directly for our simulation, any artificial modelling of the traffic loads is required.

(6) One of the most important problems of the use of the finite element method is the modelling of the structure. Especially, the structure with cracks changes its boundaries due to the growth of the cracks, and the finite element modelling becomes complex. Then, good mesh generation method is necessary for the simulation.

(7) For the modelling of a structure with one crack the size of meshes surrounding the crack is already surveyed, but for multiple cracks problem we have no information of the size of element.

3.2 Tools of the Finite Element Method

(1) Multi-level Structural Analysis 2)

In order to solve large-scale structural problem effectively and accurately, we can introduce the technique of so-called multi-level structural analysis method, i.e. Zooming Technique.(See Fig.1)

By considering the memory-size and the structure to be solved we firstly give a finite element mesh model of the total structure. Let the equations be as following;

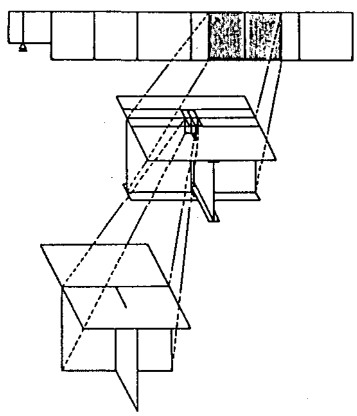


Fig.1 Zooming Technique

$$\begin{Bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{Bmatrix} \cdot \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix} \quad (1)$$

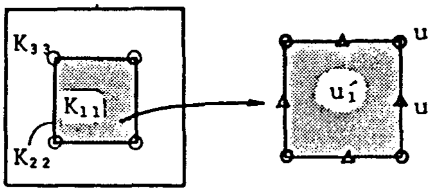


Fig.2 Variables at Zooming

, where K, u, and p are the stiffness matrix, the displacement vestor, and the force vector, respectively. And, we solve above equation for u.

Here, we consider to reanalyze the portion of the structure as shown in Fig.2. Then, eq.1 is newly expressed as following:

$$K_{11} \cdot u_1 = P_1 - K_{12} \cdot u_2 - K_{13} \cdot u_3 \quad (2)$$

We take out the portion and newly make finner mesh model for the portion. Then, we can obtain a set of linear algebraic equations as shown in (3).

$$K' \cdot u' = P' \quad (3)$$

Since the boundary of this substructure is equal to a portion of the original structure, the displacement vector of nodes surrounding the area can be determined from the solution of eq.1. That is,

$$K'_{11} \cdot u'_1 = P'_1 - K'_{12} \cdot u'_2 - K'_{13} \cdot u'_3 \quad (4)$$

Here, we recognize the difference of the number of nodes on the boundary, because the zoomed-up area has finner meshes and on its boundary more nodes are set comparing to the original one. In order to determine all values of u'-vector by using u, we introduce 2nd order Spline function for the interpolation of u'.
This procedure is repeated until we obtain a domain which is treated for the crack propagation analysis.

(2) Calculation of Stress Intensity Factors

The behaviour of cracks is wholly determined by the stress intensity factors K(I) and K(II) in case of 2-dim. problem. This indicates these values must be accurately determined, and methods having been proposed can be classified into two groups; the stress method and the dispalcement method. By considering the analysis method, the accuracy and the applicability, we introduce the latter one for our simulation method.

General displacement method for the stress intensity factors are expressed as

$$K_I = \sqrt{\frac{2\pi}{L}} \cdot \frac{G}{\kappa+1} (v'_C - v'_E)$$
$$K_{II} = \sqrt{\frac{2\pi}{L}} \cdot \frac{G}{\kappa+1} (u'_C - u'_E) \qquad (5)$$

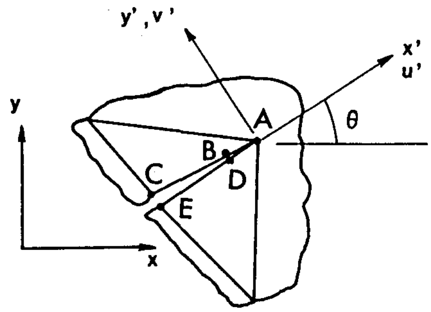


Fig.3 Crack-tip Elements

, where κ is $(3-4\nu)$ for plane strain and $(3-\nu)/(1+\nu)$ for plane stress, and L is the mesh size at the crack tip. See Fig.3 for all terminologies in eq.5.

Another effective method is proposed by Ingraffea 3), and it is expressed as

$$K_I = \sqrt{\frac{2\pi}{L}} \cdot \frac{G}{\kappa+1} [4(v'_B - v'_D) + v'_E - v'_C]$$
$$K_{II} = \sqrt{\frac{2\pi}{L}} \cdot \frac{G}{\kappa+1} [4(u'_B - u'_D) + u'_E - u'_C] \qquad (6)$$

This equation can give better result comparing with the ones by eq.5, and we introduce eq.6 into our simulation method. But, for its introduction all elements surrounding the crack-tip must be replaced by singular isoparametric elements. 4)

The characteristics of this special element can be found at the setting of the location of edge nodes, which are placed at 1/4 of the edge length instead of the center of the edge, and by this node setting the singularity of the stress distribution can be well expressed. Details of this elements are explained in successive section.

(3) Judgement of Crack Propagation

It is wellknown that any crack begins to propagate when the stress at the crack-tip reaches at a specified stress level. In fracture mechanics the stress intensity factors are used in stead of the stress, and these values are noted as the threshould, K(th). In our system K(th)'s are determined from the structural experiments.

(4) Determination of the direction and the increment of a crack

At the analysis of the crack propagation behaviour the direction and the increment of the crack length must be determined at each analysis. Obviously, these two values are wholly determined by the stress distribution at the crack-tip.

For the determination of the crack direction we introduce following two critera. 5)

(a) The maximum circumferential tensile stress criterion

(b) The minimum strain density criterion

These two criteria can determine the direction from the stress state before the crack propagates. The difference of the calculated values of these two methods are examined through the numerical experiments, and we find little difference between them.

On the determination of the increment of the crack-growth, we can find several tools all of which are empirical expressions, and they are originally from the Paris' Rule which can be expressed as following;

$$\frac{da}{dN} = C(\Delta K)^m \qquad (7)$$

, where a and N are the crack length and the number of repeated loads, respectively. C and m are the material constants of the specimen, respectively, and ΔK is the stress intensity factor width.

From this expression we can remark that the behaviour is sensitive not only to the material constants but also to the stress intensity factor, and, therefore, we have to give good values for the constants and also we have to calculate the behaviour of the crack accurately. Otherwise, the accumulation of small difference of these values leads to big difference.

(5) Treatment of Traffic Loads

Since the traffic loads are applied repeatedly to the structure and its intensity varies randomly, they must be modeled for our use. At present there does not exist any established method for the

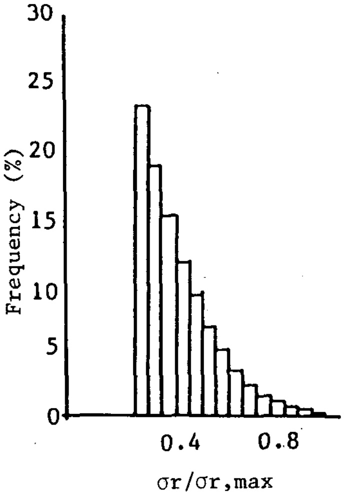


Fig.4 Blocking Loads

SA \ SS	5	10	25	50	σeq
5	78.9	91.0	97.0	97.6	98.8
10	79.5	92.2	98.2	98.8	99.4
25	80.1	92.3	98.2	98.8	100.0
50	80.1	92.3	98.2	98.8	100.0

Table 1 Accuracy of Stress Intensity Factors (%)

SS : Number of Blocks
SA : Number of Analyses for the Same Crack Length

modelling. Then, we propose following method to model the trafic loads. 5)

The data obtained from the traffic survey is divided into a number of blocks as shown in Fig.4, and we can obtain the relation between the intensity of the loads and its frequency.

Assume that several cracks locate in a structure, and solve the stress intensity factors due to unit load applied to the structure. By multiplying the actual load intensity to the calculated one, we obtain the actual stress intensity factors.

Then, calculate following eqation by introducing the stress intensity factors, and we obtain the stress intensity factor width which can be directly introduced into the Paris' rule.

$$\Delta K = \left(\frac{\Sigma n_i (\Delta\sigma_i \cdot \Delta K_{\sigma=1})^m}{\Sigma n_i} \right)^{1/m} \tag{8}$$

The accuracy of the stress intesity factor width by above method is seriously influenced by the number of blocks, and one numerical example of the application above method is shown in Table 1. According to this example the number of blocks should be more than 10.

(7) Automatic Mesh Generation Method

The macro-flowchart of the crack propagation analysis is given in Fig.5. According to this flowchart three types of mesh generation methods are required for our purpose. For the zooming technique 3-D and 2-D mesh generation methods are necessary, and also another mesh generation is required for the crack propagation analysis.

The geometry of the steel structures is complex, and the zooming technique requires the recognition of boundary nodes. Furthermore, the region where cracks propagate is small comparing to other domain, and the region where the finite element meshes must be modified is restricted in the small zone. Considering these items, we design all mesh generation methods by using the blocking method. See Fig.6.

Blocking Method is explained as following: Divide the object with complex geometry into a number of subdomains with simple geometry, divide each subdomain into finite elements, and connect them so that they form the original geometry.

This method is easily applied to three types of mesh generators required for our analysis. Especially, the former two are almost same, and 2-D mesh generator becomes the basic tool for the 3-D

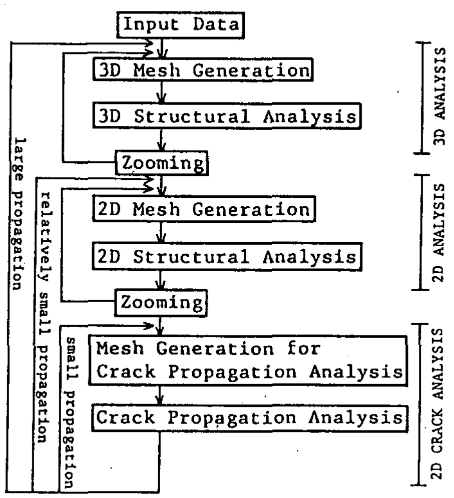


Fig.5 Crack Propagation Analysis

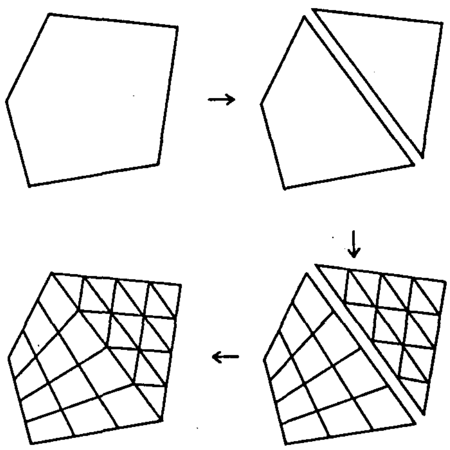


Fig.6 Blocking Method

mesh generator, because the structure treated in this investigation is formed by a gathering of plate-like structural components. Then, each structural component is divided into finite elements by using the 2-D mesh generator, and it is connected to other structural components by considering the plane where the component locates. One example of the mesh generation process of these two mesh generations is shown in Fig.7.

Assume that the area for the crack propagation analysis is obtained by the application of the zooming technique. Then, the

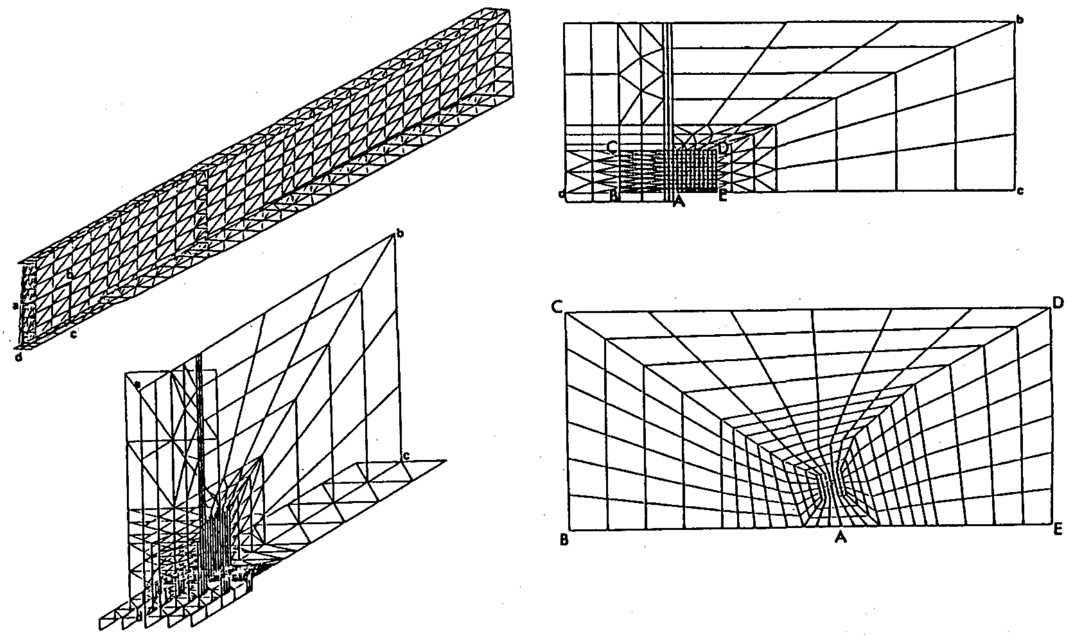


Fig.7 An Example of 2-D and 3-D Mesh Generation for Zooming Process

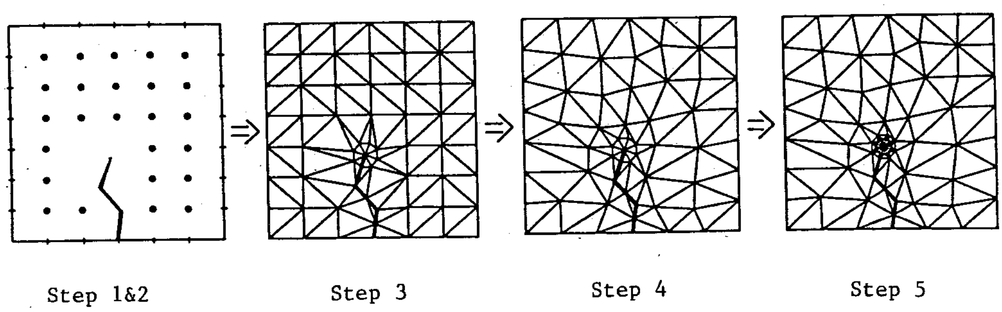


Fig.8 Mesh Generation Process using Delaunay Triangulation

geometrical configuration of the area can be assumed as simple as triangle or quadrilateral. Here, we introduce the Delaunay Triangulation Method for this mesh generation.6) We assume that the number of nodes on all edges and the location of cracks are given beforehand. The mesh generation method is shown as following:7)

Step 1 Set all nodes in the domain.
Step 2 Set more nodes on the lines where cracks locate.
Step 3 Apply the Delaunay Triangulation Method.
Step 4 Modify the location of nodes by using the Laplacian Method.
Step 5 Embed fine mesh pattern at the crack-tips.

The flow of this algorithm is illustrated in Fig.8. Refer to 7) on the details of this mesh generation method.

4. NUMERICAL EXAMPLES OF CRACK PROPAGATION SIMULATION

In this chapter we show some results of the application of the proposed method, and through these numerical experiments we show some important items and informations which are necessary for the actual numerical simulation of the crack propagation behaviour, for example the mesh size at the crack-tip area, the mesh arrangement at the area, the characteristics of the finite elements, and so on.

4.1 Three Point Bending Problem 8)

The test problem shown in Fig.9 is used for the examination of items which give us necessary informations for the setting of the finite element model. In following section we use following parameters, i.e. L, a, and w for the smallest mesh size, the crack length, and the width of the specimen, respectively.

(1) Finite Element Modelling of Crack Tip Area

The meshes near the crack-tip gives direct influence to the stress distribution, and the purpose of this section is to survey the most preferable mesh arrangement. The stress intensity factor is used for the judgement of the numerical results, and they are compared with the value obtained by the theoretical method.

Under the condition of $L/a = 0.1$, we examine the stress inten

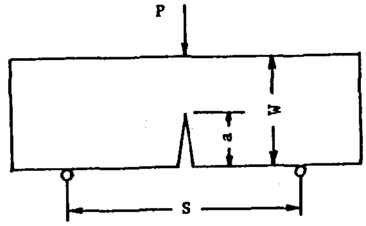
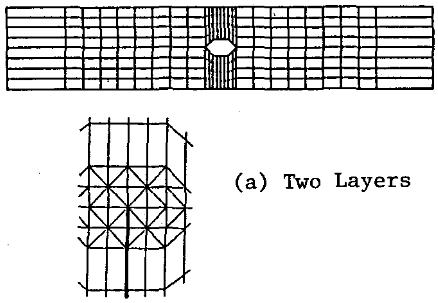
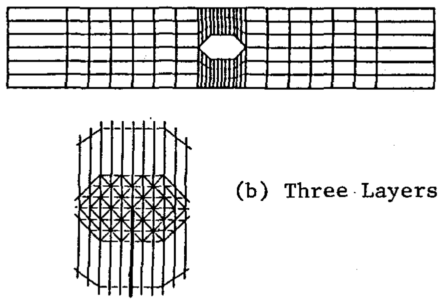


Fig. 9 Three Point Bending Problem



(a) Two Layers



(b) Three Layers

Fig. 10 Finite Element Modelling of Crack-tip Area

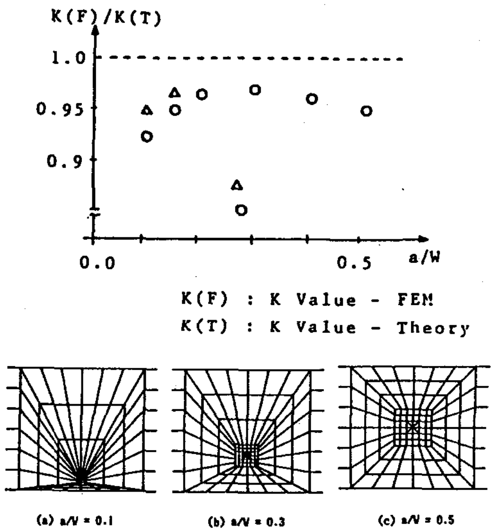


Fig. 11 Influence of a/w to K

Number of Element	K by Displ. Method	K by J-Int.
2	62.7	98.2
3	94.9	103.5

$K : K_I^{FEM} / K_I^{Theory}$ (%)

Table 2 Comparison of K's for Fig. 10

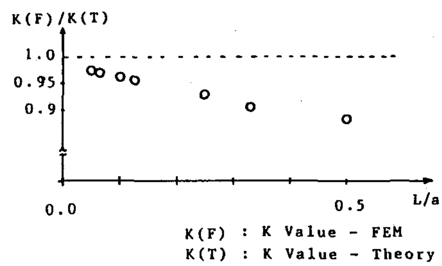


Fig. 12 Influence of L/a to K

sity factors for the meshes shown in Fig.10. The evaluation of K's is done by using eq.6.

The results are summarized in Table 2, and we can summarize that three layers of regular fine meshes should be set so that they surround the crack tip. This result is applied for successive numerical experiments.

(2) The Element Size at The Crack Tip

In this section we survey the minimum size of finite elements which are set around the crack tip. For this purpose we introduce the parameter L/a, which is the ratio of the mesh size and the crack length. The test problem is the three point bending problem of previous section, and the result obtained in former section is introduced.

We calculate the stress intensity factors for L/a = 1/3, 1/5, 1/10, 1/15, and 1/20 by using the displacement method proposed by Ingraffea. The results are summarized in Fig. 11, and it clarifies that for $L/a < 0.1$ we can obtain good results for K.

(3) The Crack Length in Specimen

This section is used for surveying the influence of a/w to the stress intensity factors. If the crack length is very small comparing to the width of the specimen, its finite element modelling becomes difficult by the reason of the limited CPU memory. Then, the value a/w becomes an important parameter for the modelling.

The results of numerical experiments are summarized in Fig.12, and from it we can conclude that a/w should not be set too small and also too large. This is caused by the existence of strained meshes, and therefore, if we are allowed to use more memory, then this restriction can be removed.

4.2 A Plate Structure with a Manhole

All the results obtained in previous section is introduced in the modelling of a plate with a manhole shown in Fig.13.

For the analysis we introduce the zooming technique, and we obtain the peak principal stress at the corner of the manhole. See Fig.14. From this result we set the initial crack of length 10mm at the position, and we continue the crack propagation analysis till

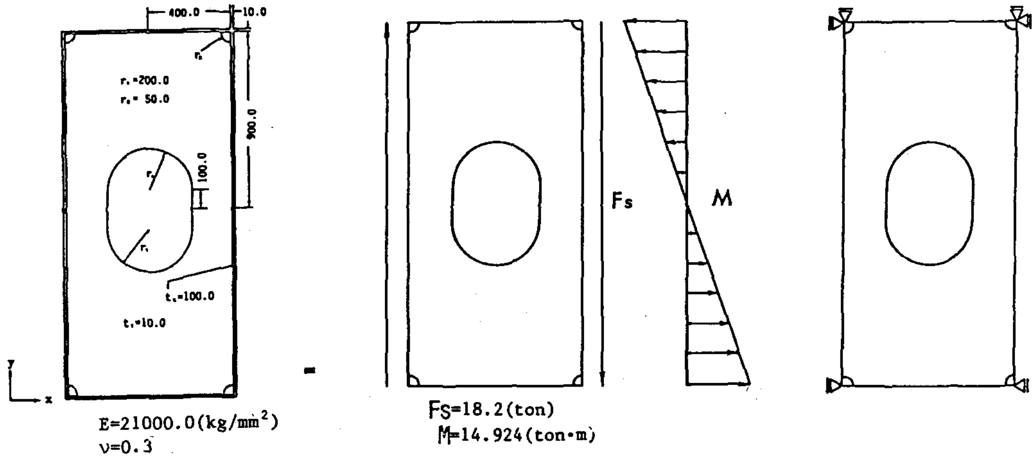


Fig.13 A Plate with a Manhole: Geometry and Boundary Conditions

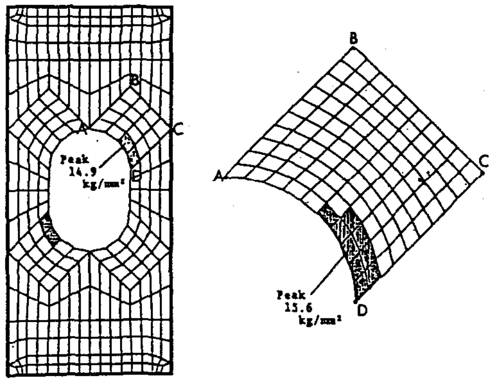


Fig.14 Model and Result

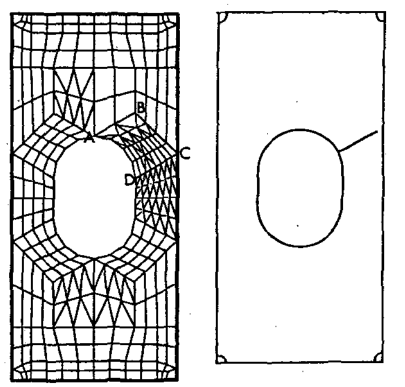


Fig.15 Result of Crack Propagation Analysis

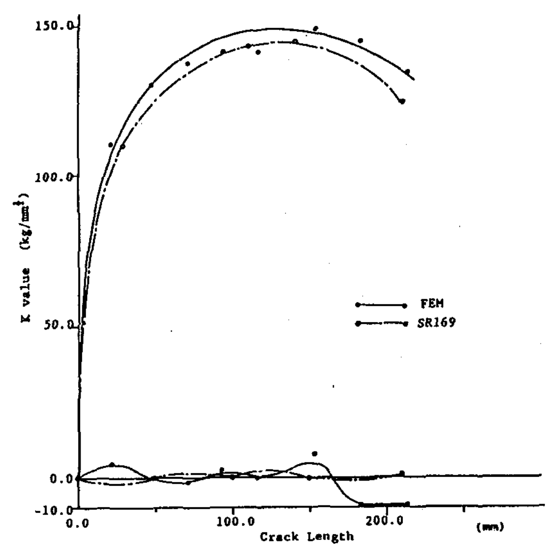


Fig.16 Stress Intensity Factors

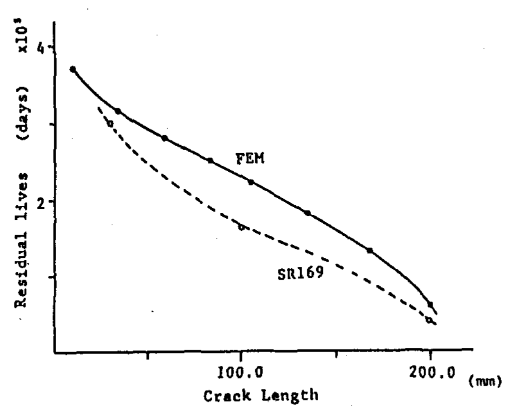


Fig.17 Residual Life

the crack length reaches at 270mm. The analysis is repeated for 8 times. See Fig.15.

The computational results are compared with the ones given in Ref.9. The comparison is done by using the stress intensity factors at each analysis step, and also the residual life of the structure. (See Fig's 16 and 17.) The results show that they show good agreements each other.

4.3 Multi-crack Propagation Analysis

A portion of steel structure shown in Fig.18 is used for the test problem of the numerical simulation of the multi-crack propagation. In this structure five pieces of plate are welded, and by considering its symmetricity a half of it is used for the analysis as shown in Fig.19. From the structural experiment we knew that fatigue cracks occure from two not-welded part of the figure, and henceforce we call them Cracks A and B, respectively. Furthermore, the structural experiments clarify that Crack A grows lately than Crack B, but the final collapse occures by the growth of the crack A. Then, our purpose is to reappear this phenomenon. The boundary condition and the loading condition are given in the figure.

From the structural experiments we can forecast that the crack A changes its growth direction upward at the beginning, and for the reappaerance of this curved crack very fine meshes must be set at the portion. For this examination we set two types of mesh sizes, i.e. 0.25mm and 0.025mm as the smallest mesh size at the crack tip 10). At the numerical experiment we calculate the number of loadings which is required for the prescibed crack growth, i.e. 0.25 or 0.025 mm, for the crack with smaller stress intensity factor.

The results are illustrated in Fig's 20 and 21, and the details are summarized in Table 3. The results by the fatigue tests are shown in Table 4. From these results we find that the simulation using the coarse meshes cann't reappear the actual phenomenon, but the result of the finer meshes can show good coincidence with the fatigue test. From these test we can conclude that if multiple cracks are treated, ten times finer meshes are required comparing to the single crack propagation analysis.

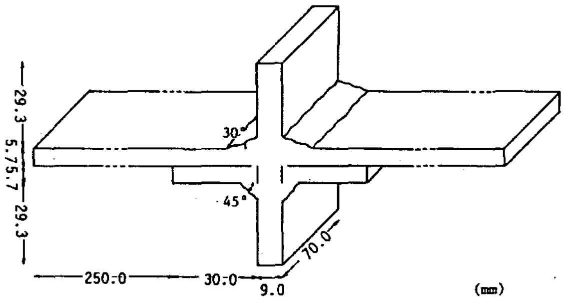


Fig.18 Geometry of Structure

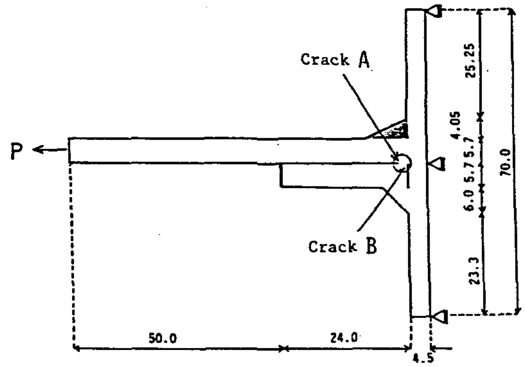


Fig.19 Geometry of Structure for Analysis

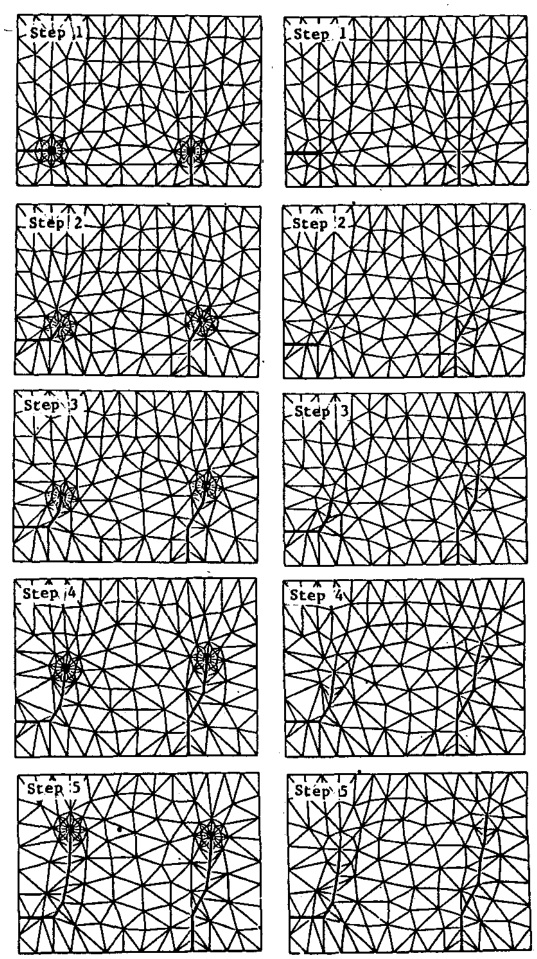


Fig.20 Crack Propagation Behaviour
Left: Fine Mesh, Right: Corase Mesh

Table 3 Calculated K Values
Upper: Fine Mesh
Lower: Coarse Mesh

	Cycles	K·Crack A	K·Crack B
Step 1		14.8546	18.0165
Step 2	44*10⁴	19.1889(0.25)	20.2397(0.42)
Step 3	62*10⁴	22.1765(0.50)	21.7703(0.68)
Step 4	77*10⁴	26.0743(0.81)	23.2053(0.93)
Step 5	91*10⁴	33.4470(1.29)	23.5897(1.18)
Step 6	98*10⁴	37.0881(1.72)	23.2001(1.30)

K·(kgf/mm$^{3/2}$) ( ) Crack Length(mm)

	Cycles	K·Crack A	K·Crack B
Step 1		13.2470	18.5728
Step 2	44*10⁴	19.3090(0.25)	21.5207(0.50)
Step 3	64*10⁴	21.7576(0.50)	23.9165(0.85)
Step 4	79*10⁴	24.5289(0.75)	26.5217(1.18)
Step 5	89*10⁴	27.2073(1.00)	29.2442(1.49)

K·(kgf/mm$^{3/2}$) ( ) Crack Length(mm)

Table 4 Results of Fatigue Test

No.	$\Delta\sigma$ kg/mm^2	Collapse Occured at	Propagation Times (Cycles)
P1	12.1	Crack A	1,059,500
P2	15.1	Crack A	586,500
P3	18.0	Crack A	287,800

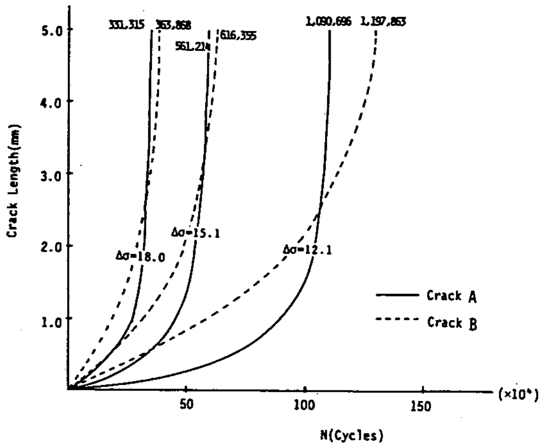


Fig.21 Crack Propagation Behaviours (Fine Mesh)

5. CONCLUDING REMARKS

In this investigation the authors explained the numerical simulation method of single and multiple cracks propagation in steel structures. Moreover, through numerical experiments they showed some useful informations for the actual simulation technique.

Through the numerical experiments they could show that the system given in this paper can reappear single and multiple cracks propagation phenomena by using the computer. This indicates that the system may become an efficient and powerful tool for the estimation or the forecasting of the residual lives of existing steel structures with cracks. At the same time, it is obvious that the system can easily give , for example, the boundary condition or the loading condition for the fatigue test which treats only the substructure and also it can explain the details of the structural experiments.

REFFERENCES

1) M.Shiratori, T.Miyoshi, & H.Matsushita,'Numerical Fracture Mechanics', Jikyou-shuppan, 1980 (In Japanese)

2) K.Moriwaki & T.Taniguchi,"Numerical Experiments on FE Modelling for Multi-level Structural Analysis",J. Structural Eng., Vol.35A, pp.223-230 1989 (In Japanese)

3) A.R.Ingraffea,"Numerical Modelling of Fracture Propagation", Rock Fracture Mechanics(ed. by H.P.Rossmaith), Springer Verlag, pp.151-208, 1983

4) B.S.Barsoum,"On the use of Isoparametric Finite Elements in Linear Fracture Mechanics", Int. J. Num. Meth. in Eng.,Vol.10, pp.25-37, 1976

5) K.Sanada,'Fundamental Study on the Estimation of the Residual Life of Steel Structures for Fatigue Crack',Master Thesis in Okayama University, 1988

6) S.W.Sloan,"A Fast Algorithm for Constructing Delaunay Triangulations in the Plane", Adv. Eng. Software, Vol.9, No.1, 1987

7) C.Ohta & T.Taniguchi,"Automatic Mesh Generation Method for Multiple Crack Propagation Analysis", Annual Conference of Chugoku-Shikoku Branch, JSCE, pp.10-11, 1989

8) T.Kunio et al,'Experimental Method for Fracture Mechanics', Asakura, pp.241-242, 1984 (In Japanese)

9) The 169 Working Group in Japan Association of Ship Building,'Design Guide of Fracture Controle of Ship Structure',1983 (In Japanese)

10) K.Washizu et al,'Finite Element Method Handbook', Vol.2, Baihuhkan, pp.305-422, 1983 (In Japanese)