

Zeitschrift: IABSE reports of the working commissions = Rapports des commissions de travail AIPC = IVBH Berichte der Arbeitskommissionen

Band: 34 (1981)

Artikel: Man-machine interactive analysis in concrete structures using a microcomputer

Autor: Maekawa, K. / Yamazaki, J. / Sakurai, T.

DOI: <https://doi.org/10.5169/seals-26900>

Nutzungsbedingungen

Die ETH-Bibliothek ist die Anbieterin der digitalisierten Zeitschriften auf E-Periodica. Sie besitzt keine Urheberrechte an den Zeitschriften und ist nicht verantwortlich für deren Inhalte. Die Rechte liegen in der Regel bei den Herausgebern beziehungsweise den externen Rechteinhabern. Das Veröffentlichen von Bildern in Print- und Online-Publikationen sowie auf Social Media-Kanälen oder Webseiten ist nur mit vorheriger Genehmigung der Rechteinhaber erlaubt. [Mehr erfahren](#)

Conditions d'utilisation

L'ETH Library est le fournisseur des revues numérisées. Elle ne détient aucun droit d'auteur sur les revues et n'est pas responsable de leur contenu. En règle générale, les droits sont détenus par les éditeurs ou les détenteurs de droits externes. La reproduction d'images dans des publications imprimées ou en ligne ainsi que sur des canaux de médias sociaux ou des sites web n'est autorisée qu'avec l'accord préalable des détenteurs des droits. [En savoir plus](#)

Terms of use

The ETH Library is the provider of the digitised journals. It does not own any copyrights to the journals and is not responsible for their content. The rights usually lie with the publishers or the external rights holders. Publishing images in print and online publications, as well as on social media channels or websites, is only permitted with the prior consent of the rights holders. [Find out more](#)

Download PDF: 11.12.2025

ETH-Bibliothek Zürich, E-Periodica, <https://www.e-periodica.ch>

Man-Machine Interactive Analysis in Concrete Structures Using a Microcomputer

Une analyse interactive des structures en béton utilisant un micro-ordinateur

Mann-Maschine-Interaktionsanalyse von Stahlbeton mithilfe von Microcomputern

K. MAEKAWA

Graduate Student
University of Tokyo
Tokyo, Japan

J. YAMAZAKI

Associate Professor
Tokyo Metropolitan University
Tokyo, Japan

T. SAKURAI

Advanced Numerical
Analysis Inc.
Tokyo, Japan

SUMMARY An interactive mode of operation using a microcomputer for nonlinear finite element analysis is presented. An example of how nonlinear phenomena can be analyzed using an interactive mode of computation is presented. The example predicts the nonlinear formation of cracks in a loaded concrete block. The calculated results closely resemble actual behaviour.

RÉSUMÉ

Cette étude présente une méthode d'opération interactive pour l'analyse non-linéaire par éléments finis au moyen d'un micro-ordinateur. L'exemple d'application prédit la formation de fissures dans un bloc en béton non-armé soumis à une charge uniaxiale. Les résultats calculés correspondent au comportement réel observé.

ZUSAMMENFASSUNG

Eine interaktive Methode für nichtlineare finite Elementeberechnungen mit einem Microcomputer ist gezeigt und auf das Beispiel der nichtlinearen Rissbildung in einem Betonblock angewandt. Die Rechenergebnisse stimmen mit dem wirklichen Verhalten gut überein.



1. INTRODUCTION

Recent and drastic developments of electronics have made inexpensive, but high performance, microprocessors available. This has made it possible for individuals to control the whole process of computer computation that previously could be handled only by large computers that were not readily accessible. This trend of using microcomputers for personal computation implies a possibility that the present inconvenient and cumbersome nature typical of analysis for concrete mechanics may become manageable and practicable. This paper discusses a "personal" method of analysis with reference to an example of numerical analysis. In the near future microcomputers will become a common tool for engineers. When compared to large computers, the microcomputers obviously suffer limitations in terms of computation abilities, various functions, and memory sizes. But on the other hand, because the individual users handle all-operations it might be more practical than present methods of analysis using large computers with the inconvenience of poor turn around time. The current limitation in computation ability of microcomputers will be reduced when the sixteen bit CPU that are already in existence become more generally available. The numerical computations described in this paper were performed on an 8-bit CPU microcomputer. The finite element program used is called FEMN (Finite Element Method of analysis for Nonlinear problems) and was developed by Sakurai as a part of the development work for NAPRA (Nonlinear Analysis Program Research Association). Several reports have been published on application of microcomputers for finite element methods of analysis, which include works by Wilson [1] and Yamada [2].

2. HARDWARE OF MICROCOMPUTER SYSTEM

A microcomputer system for analysis is shown in Fig.1. The basic requirements are a CPU with a memory unit, a display, a keyboard, an extended memory and a printer. A plotter or other graphics equipment is desirable. The memory required is about 64 k bytes of RAM and about 1 M bytes for the extended memory for finite element analyses. The capacity of this system can be expanded by connecting it to a large size computer through a sonic coupler and communication line.

3. CHARACTERISTICS OF MICROCOMPUTERS

The followings are meritorious features of microcomputers when compared to common usage of large computers.

3.1 Creation of algorithms

The personalized computers can be used exclusive by an individual or a small group of people. This makes it convenient and easy to develop programs, to debug, or to correct algorithms. The algorithms are portable when written in a common language such as FORTRAN and therefore can be run on large computers.

3.2 Economy

The major operating cost of microcomputers is for electricity. The initial cost for the system shown in Fig.1. is about 6,000 US dollars, which is affordable for a small group of people or an individual. This investment can be justified in the long term from a view point that most of the time for research work the microcomputers will be used for developing programs. The price trend is expected to continue to decrease thus further reducing the need to consider the hardware cost.

3.3 Interactive operation

The most obvious positive characteristic of computation by microcomputers is the possibility of interactive operation, which is particularly desirable, in nonlinear analysis. Interactive operation accomplished, when using microcomputers, can be by inputting various commands or data at intermediate stages of computation. By knowing the results of the previous step(s) of computation, intelligent modifications of boundary conditions or load increments for the next step(s) can be made for example, behavior of structure subject to hysteretic or time dependent loading. Thus feedback of the analyst judgement at various stages of the analysis process becomes very easy.

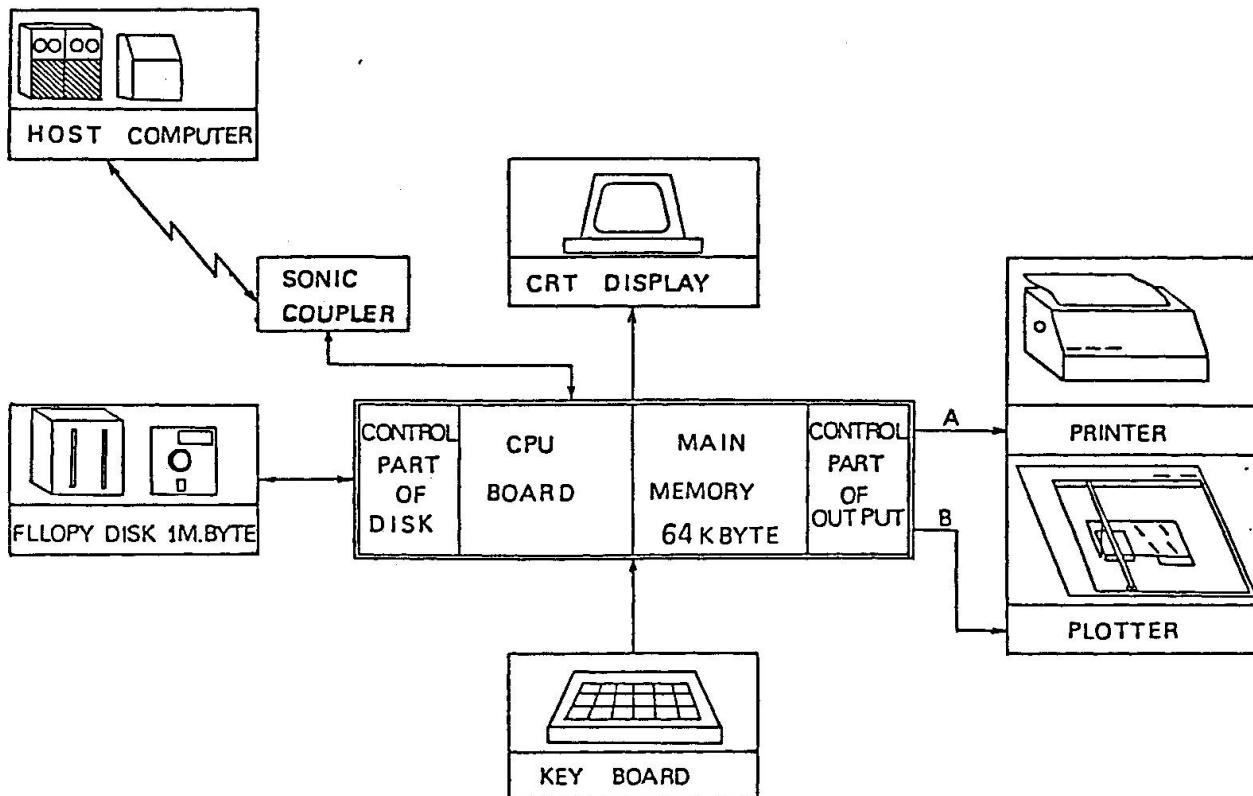


Fig.1 Microcomputer system



4. FINITE ELEMENT METHOD SOFTWARE PROGRAMMED FOR MICROCOMPUTERS

The limitation of 64 k byte of main memory dictates tight design of the main program and an efficient use of the areas for variables for computation. Dynamic allocation of the area for variables is essential if large size problems are to be solved by the same program on large computers. A one dimensional common array called "POOL" is used to perform computation and thus just by changing the dimension of this array, programs of various sizes can be run on large computers. The common array is controlled by three subprograms named OPEN, CLOSE and SEARCH. The data content in the POOL array is shown in Fig.2.

COMMON ARRAY POOL

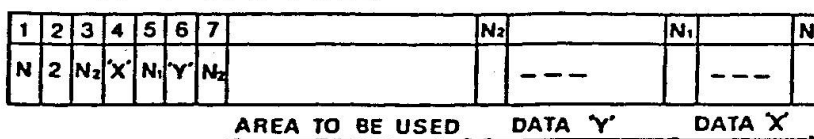


Fig.2 Constitution of common array POOL

The number of the dimension of POOL array is stored in POOL(1), and POOL(3) contains the top address of data sets which are input. The name and the address of each data set is stored starting at POOL(4) and continuing sequentially as required. The corresponding data sets are stored from the bottom of POOL. The OPEN subprogram writes the names of the arrays to be opened in the common data area, and reserves the memory area. Then the data are transferred to this area and processed. The CLOSE subprogram deletes unoccupied variables area and packs the data of the POOL, and the indices are modified as indicated in Fig.3.

COMMON ARRAY POOL

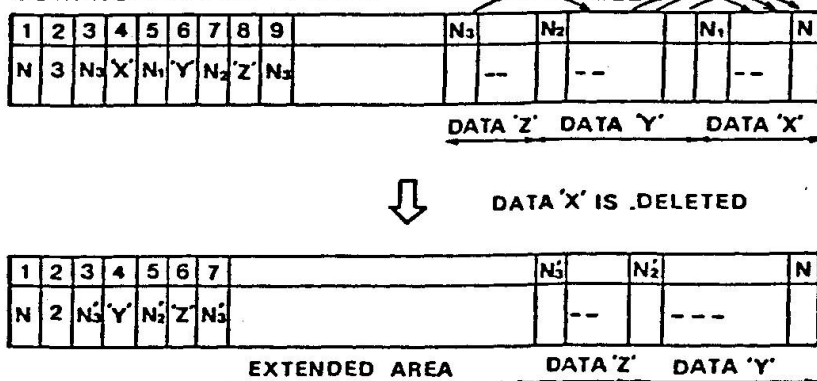


Fig.3 Function of subprogram CLOSE

The SEARCH subprogram searches to find certain data in the common area. Since the common area is controlled by these three subprograms, it is only necessary to change the array declaration of POOL to adjust for the various sizes of problems.

The solution of the equilibrium equations is by the Cholesky method incorporating the skyline technique and a modified Newton-Raphson iteration.

The stiffness matrix is processed in-core, because data transfer between the external files and core is time consuming.

5. MICROCOMPUTER CAPABILITY FOR NONLINEAR ANALYSIS OF CONCRETE MECHANICS

Utilizing the characteristics of microcomputers stated above, three capabilities were devised for analysis of concrete mechanics.

These are (1) Restart, (2) Split node, and (3) Relocatable node capabilities.

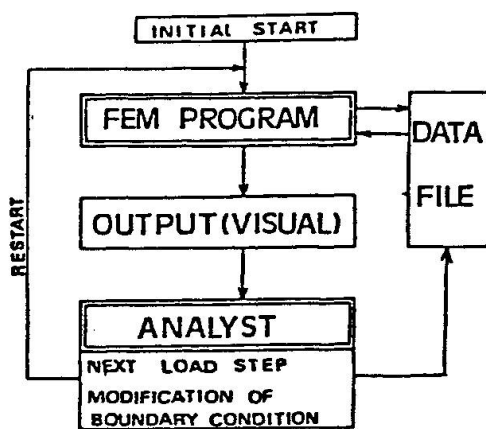


Fig.4 Flow chart of restart procedure

5.1 Restart

The "Restart" capability means the ability to suspend and restart a sequential computation, that is, the latest results of incremental computation are stored in the extended file and then the next increment of computation is started. Input is allowed at this stage for changing the boundary conditions and values and for incrementing the loads or displacements. The process is shown in Fig.4. Thus the intermediate results are preserved, at any stage and as many times as required, until an appropriate load

increment or other conditions are found by repetition of numerical simulation.

Such a process is useful and conveniently performed by the interactive handling as indicated above.

5.2 Split node

For a representation in analysis model of cracks in concrete, a formation of new surfaces or planes is more realistic than the frequently used smeared crack representation[4],[7] (altering the stiffness of a whole element) because the behavior of certain types of concrete structures is significantly influenced by formation of isolated cracks. Several reports[3],[5] have been published on the analysis of concrete structures with cracked surfaces. In this case, the analysis with initially cracked element mesh can not calculate the inner stresses or strains when the constitutive equations have parameters influenced by the stress or strain history such as plastic strain. The following formulations try to make it possible using two dimensional 3 to 8 variable node isoparametric elements.



The nodes to be split in two to separate adjacent elements representing crack formation are specified initially, and only the locations in the stiffness matrices are reserved, see Fig.5. Then, the nodes to be split are specified at the Restart, and the B matrix and stiffness matrix are formed using the coordinates before the split.

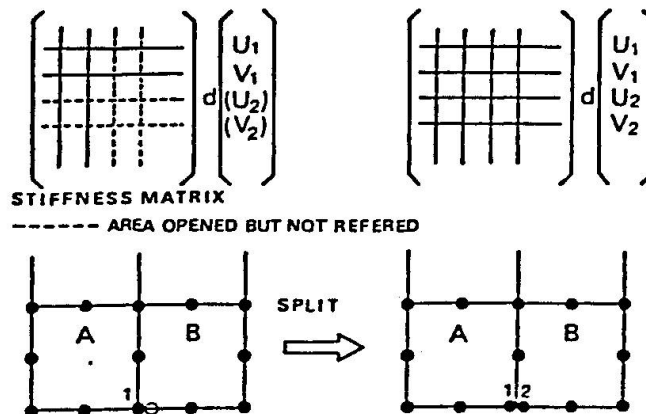


Fig.5 Split node

By this operation only the number of nodes is increased as shown in Fig.5 but not the number of elements. Then the computation is executed by letting the displacements for the split node (new node) equal to those for the node before splitting. When a crack is formed, the stresses and strains for all Gauss points in the involved elements use the values for the previous step. Hence, in general, the external and internal forces are not equilibrated and the difference will be,

$$V = F - \iiint B^T \sigma dv \quad \text{--(eq.1)}$$

where. V = unbalanced nodal force vector
 F = external nodal force vector
 B = strain displacement matrix
 σ = stress vector

The next correction of nodal displacements will be,
 $dU = K^{-1} V \quad \text{--(eq.2)}$

where dU = increment of nodal displacement vector
 K = global stiffness matrix at each step

and so,

$$U = U + dU \quad \text{--(eq.3)}$$

$$\xi = \xi + d\xi \quad \text{--(eq.4)}$$

$$\sigma = D(\xi, d\xi) \quad \text{--(eq.5)}$$

where U = total nodal displacement vector
 $\xi, d\xi$ = total, and incremental strain vector
 D = constitutive equations

The stresses (eq.5) are substituted into eq.1, and if the quantity V is not as small as required, the next iterative procedure must be executed using eq.2 through eq.5. Thus, computation is repeated until the value V becomes as small as required.

5.3 Relocation of nodes or modification of finite element mesh

When cracking is to be simulated in a model by release of connectivity of the variable node isoparametric elements, a necessity arises for the orientation of a side of elements to be altered to align with the direction of the propagation of the cracks, or the mesh layout is to be modified each time when required. Here, only geometric alteration of node positions takes place but not the number of elements. When this procedure is used the B matrices for the elements are corrected at the Restart. Furthermore, since the Gauss points for evaluation of stresses and strains are altered, these stresses and strains need to be corrected when moving the finite element mesh. Calculating the stresses and strains at the next step, these are calculated by use of the stresses and strains at the previous conditions. But, when the mesh of the finite elements are altered, there is no information at the new Gauss points so that the initial conditions of iterative procedure must be corrected.

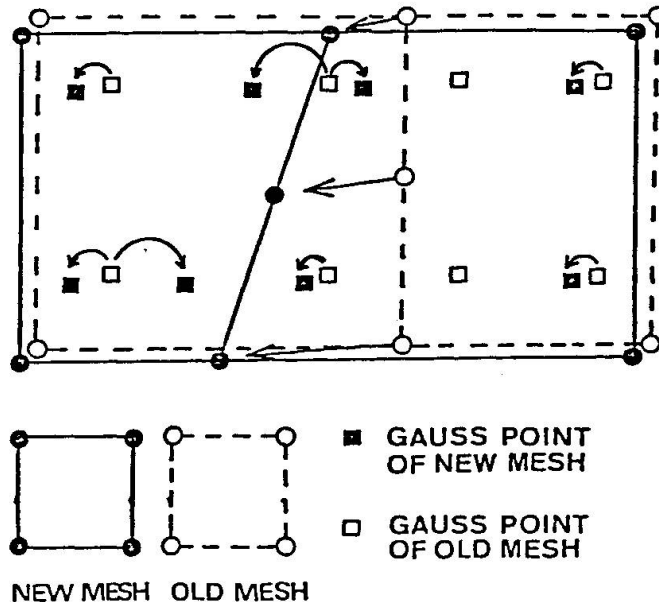


Fig.6 Correction of initial condition at each step due to relocation of finite element mesh

In this example as shown in Fig.6, correction is made by letting the nearest (in global coordinate system) Gauss points as an initial conditions based on the Gauss points in the previous computation step. Correction of nodal displacements is made by calculating the displacements at the new points of altered nodes with the interpolation functions and nodal displacements of previous computation step. If a perfect correction is made, V in eq.1 is equal to zero. But in actual case, the unbalanced nodal force vector V appears by the modification of finite element mesh, so the nonlinear trial and correction procedure (from eq.1 to eq.5) is executed, and the initial condition of iterative procedure is modified.



The flow charts in Figs. 7 and 8 indicate a method for an analysis including the relocation procedure and a method for correction of stresses and strains at Gauss points. The three procedures described above might appear cumbersome, but they can be handled because the process is sufficiently flexible and effective when installed on a personal computer systems. Such capabilities interacted with human decisions, and thus a synthesis of analyst and computer system enables a simulation of crack propagation in concrete structures which has been difficult to handle before.

The following is an analysis example.

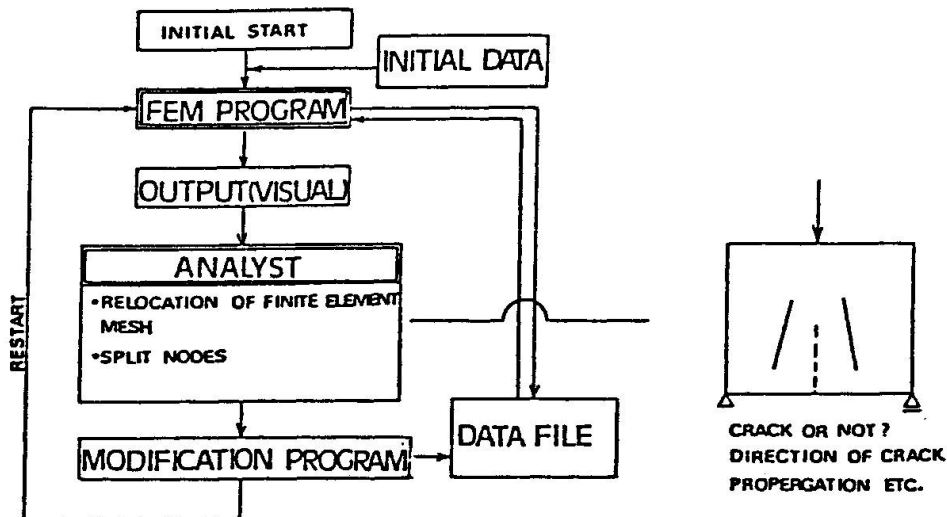


Fig.7 Flow chart of split node and relocation of finite element mesh

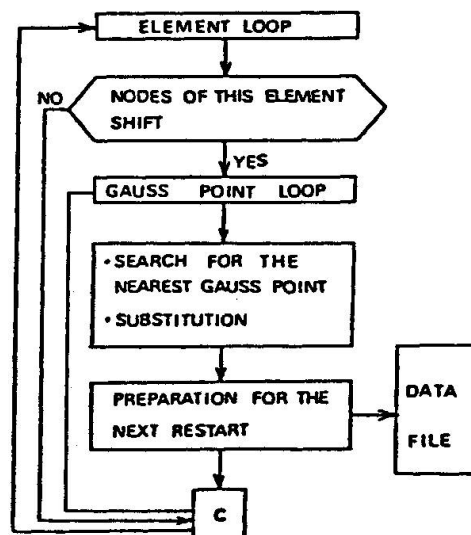


Fig.8 Modification of initial condition of iterative procedure for the next Restart

6. SAMPLE ANALYSIS

A simplified example, as reported in Ref.[6], was made that used the three computational capabilities stated above. A block of concrete was loaded at the center through a top and bottom steel plates as shown in Fig.9. The finite element mesh is shown in Fig.10. At the start of computation, all the nodes were specified to allow splitting at any node(s), and locations in the stiffness matrices were reserved in memory. The boundary conditions are indicated in Fig.10, and the top and bottom surfaces of the plate are considered to displace equally. The actual crack pattern is shown in Fig.11. The order for integration was made equal to 2. The concrete was assumed to be elastic, its modulus of elasticity E_0 was assumed to be $3.06 \times 10^4 \text{ N/mm}^2$, the poisson's ratio to be 0.2, and tensile strength F_t to be 2.55 N/mm^2 . Other assumptions are a plane stress condition and perfect contact between the steel plates and the concrete block.

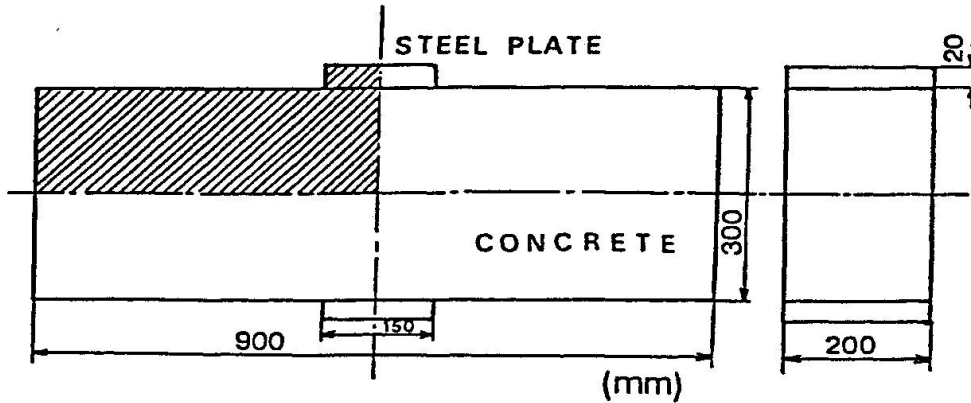


Fig.9 Concrete block

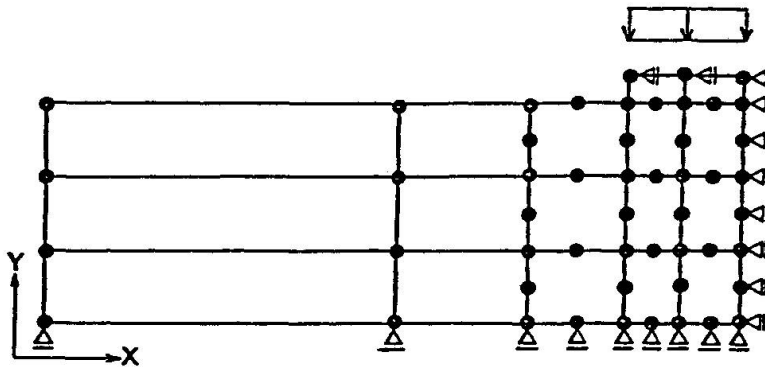


Fig.10 Initial mesh of Stage 1

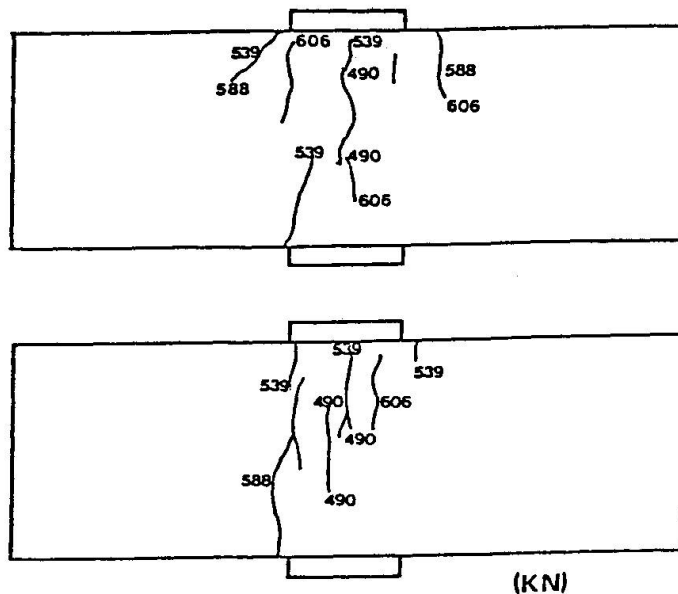


Fig.11 Actual crack pattern of a concrete block

Here, the decisions for the relocation of the mesh and the selection of nodes to be split were made at each step by the analyst interactively after a critical review of the intermediate results.

Cracking was assumed to occur at a node nearest to the point where the principal stress exceeded the tensile strength of concrete F_t . Then the node was split and the boundary condition was changed, and the mesh was altered so that the orientation was aligned to the direction of the principal stress.

Stage 1 - Stage 3 ; Fig.12-1, Fig.12-2
When the displacement of Node 100,101,102 = -0.06mm(Y-direction), the stress exceeded F_t at Gauss point A, and hence the boundary condition restraints for the nearest Node 1 was released to represent cracking at restart(Stage 1). The restart resulted in excess of F_t for the major principal stress at the point B, so that the Node 2 was made to be a free node(Stage 2). By the release of the boundary conditions a flexural crack propagated as shown in Fig.12-2.

Stage 3 - Stage 5 ; Fig.12-2, Fig.12-3
The restart resulted in excess of F_t for the major principal stress at the point C, (Stage 3) consequently, the boundary conditions for Node 3 was altered at restart(Stage 4). In the same procedure as the upper stage, the first crack was extended as shown in Fig.12-3(Stage 5).

Stage 5 -Stage 6 ; Fig.12-3, Fig.12-4

As a consequence of extension of crack, the principal stress exceeded F_t at the point D.(the increment for displacements is zero.)

Hence, Node 4 was split in Fig.12-4 and there a new node was created to indicate the second crack formation(Stage 6).

Stage 6 -Stage 7 ; Fig.12-4, Fig.12-5

With the condition of displacement increment being equal to zero, the unbalanced forces were distributed to the affected nodes.

Due to the resulting stresses, the principal stress exceeded F_t at the point E(Stage 6). Hence Node 5 was split and node 6,7 and 8 were relocated as shown in Fig.12-4.

Stage 7 - Stage 8 ; Fig.12-5, Fig.12-6

The second crack extended, and the stresses were redistributed as shown in Fig.5(Stage 7). Only the stress of point F exceeded F_t , therefore, Node 9 was split and then all major principal stresses were lower than F_t . Hence, a new displacement increment = -0.0026mm (Y-direction) was imposed to Node 100,101,102.

Stage 8 - Stage 9 ; Fig.12-6, Fig.12-7

Imposing the new displacement the major principal stress at the point G exceeded F_t (Stage 8). This time, the direction of the major principal stress was approximately parallel to the X-axis, and therefore the node was split but the mesh was unchanged as shown in Fig.12-7.

Stage 9 - Stage 10 ; Fig.12-7, Fig.12-8

By the split of Node 10, a third crack was formed. This time, all principal stresses were lower than F_t so that the displacement increment = -0.0044 (Y-direction) was imposed to Node 100,101 and 102. The resulting principal stress of points H and I exceeded F_t (Stage 9). Therefore, Node 11 was split by the same manner as Stage 8 and 9.

Stage 10 ; Fig.12-8

The third crack was made using the function of split node and the stresses were redistributed as shown in Fig.12-8, and the reaction at the plate was decreased(Stage 10). The relationship between the displacement at each stage and the reaction is shown in Fig.13.



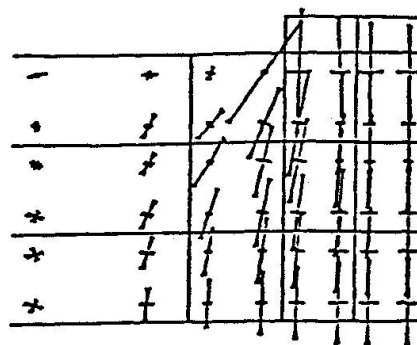
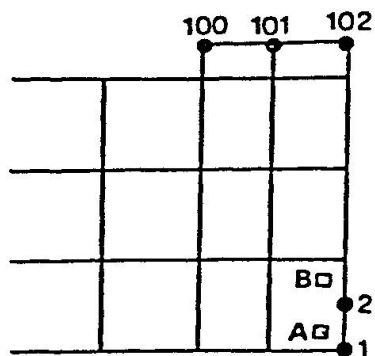
STAGE 1

 $U = -0.06$ $R = 496$

where

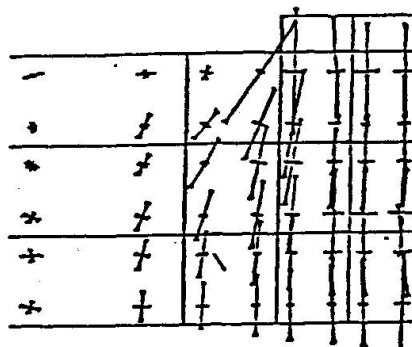
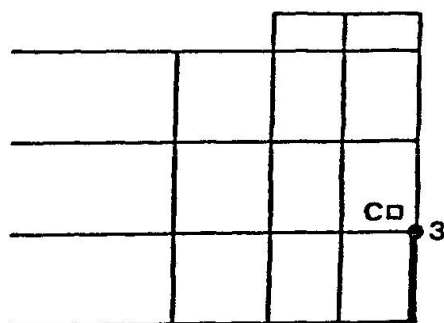
U ; DISPLACEMENT OF
NODE 100-102(MM)

R ; LOAD(KN)



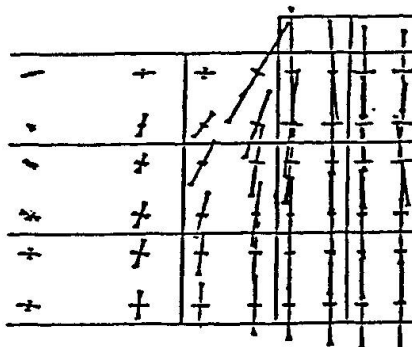
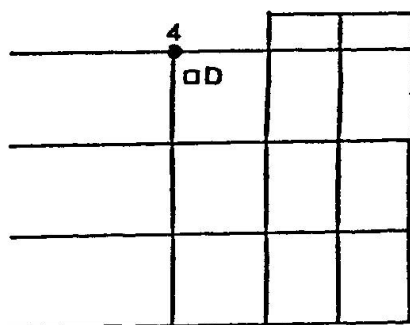
[Fig.12-1]

STAGE 3

 $U = -0.06$ $R = 492$ 

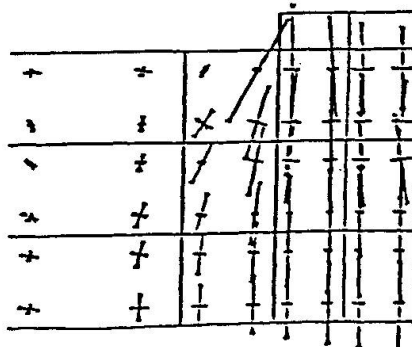
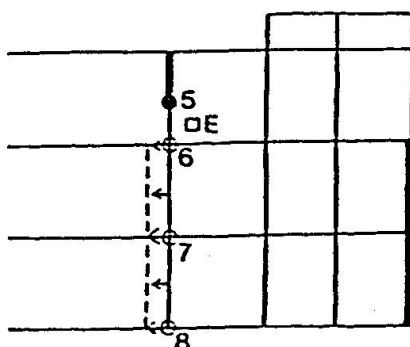
[Fig.12-2]

STAGE 5

 $U = -0.06$ $R = 479$ 

[Fig.12-3]

STAGE 6

 $U = -0.06$ $R = 475$ 

[Fig.12-4]

STAGE 7

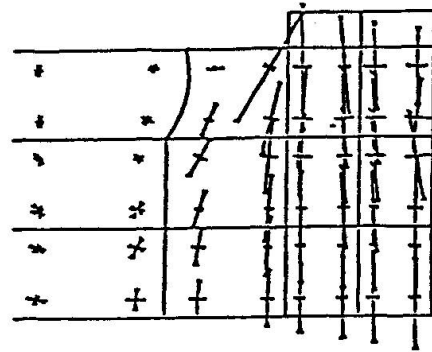
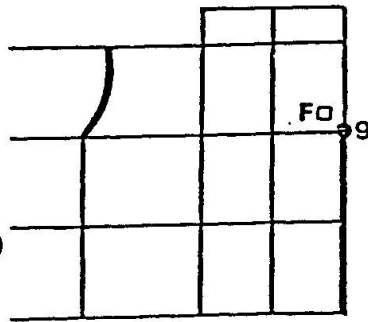
$U = -0.06$

$R = 473$

where

U ; DISPLACEMENT OF
NODE 100-102(MM)

R ; LOAD(KN)

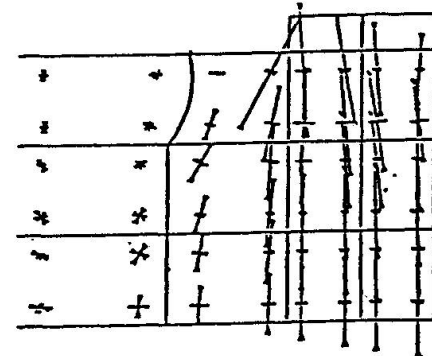
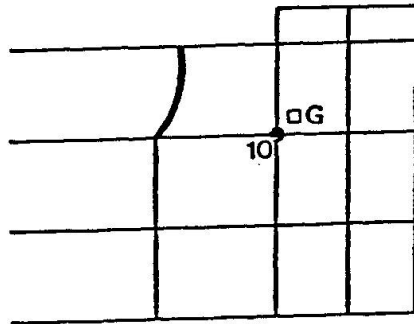


[Fig.12-5]

STAGE 8

$U = -0.0626$

$R = 487$

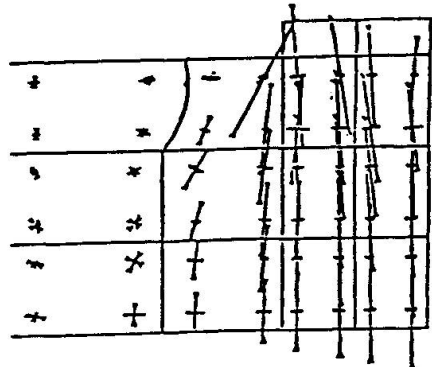
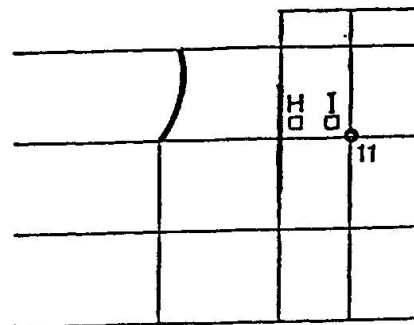


[Fig.12-6]

STAGE 9

$U = -0.067$

$R = 519$

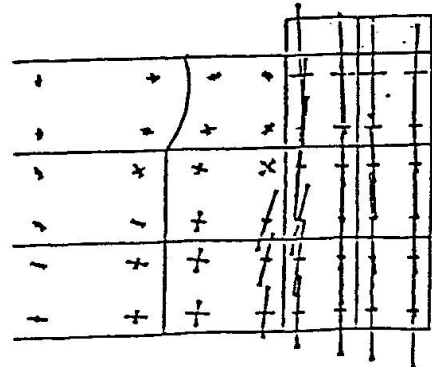
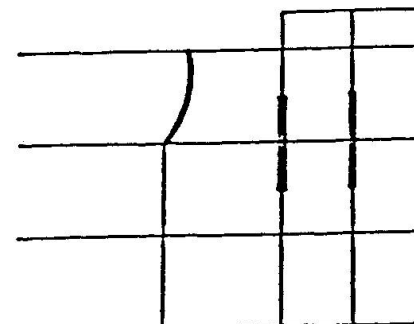


[Fig.12-7]

STAGE 10

$U = -0.067$

$R = 427$



[Fig.12-8]

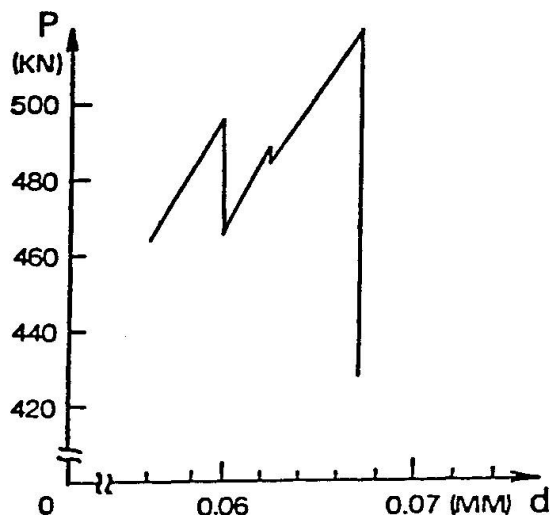


Fig.13 Calculated load-displacement relationship

7. CONCLUDING REMARKS

Because of the convenience of using microcomputers which are rapidly becoming available to all engineers, the methods of computation, which are different from those typically used for processing on large computers, appear to be undergoing change. In the near future it will be possible for engineers to analyze complex concrete mechanics problems at their convenience at their home or desk.

The possibility for improvement through new methods was demonstrated by an example of analysis and computer system presented in this paper. The improvement was made possible because the engineer was able to use the microcomputer in an interactive mode.



REFERENCES

1. E.L. Wilson, "SAP-80 Structural Analysis Programs for small or large Computer Systems"
2. Y.Yamada, H.Okumura and T.Sakurai, "System for Medium-Sized and Experimental Finite-Element Analysis" presented at Symposium of Computational Methods in Nonlinear Structural and Solid Mechanics, Oct. 6-8, 1980, NASA.
3. A.C.Scordelis, D.Ngo and H.A.Franklin, "Finite Element Study of Reinforced Concrete Beams with Diagonal Tension Cracks" ACI SP 42-2.
4. D.Darwin and D.A.Pecknold, "Nonlinear Biaxial Stress Strain Law for Concrete", ASCE Journal, Engineering Mechanics Division EM2 April 1977
5. A.H.Nilson, "Nonlinear Analysis of Reinforced Concrete by the Finite Element Method", ACI journal Sept. 1968.
6. A.C.T.Chen and W.Chen, "Constitutive Equations and Punch-Indentation of Concrete", ASCE Journal, Engineering Mechanics Division EM6 December 1975.
7. E.Raue, "Limit Analysis of Reinforced Concrete Shells of Revolution", IABSE Colloquium Copenhagen Final Report 1979

Leere Seite
Blank page
Page vide