Memoirs of the School of Engineering, Okayama University Vol. 20, No. 2, February 1986

Two-Dimensional Automatic Mesh Generator for Finite Element Analysis

TAKEO TANIGUCHI*, KAZUHIKO MITSUOKA* & EIJI KAJITANI**

(Received February 17, 1986)

SYNOPSIS

In this study a new automatic mesh generator for 2-dimensional finite element analysis is proposed, and its effectivity is surveyed through a number of test examples. Proposed one is for a micro-computer, and the program is written in BASIC. The user needs no preparation for making finite element model in advance. All of the neccessary informations are displayed on CRT display and its user may answer for questions. It is expected that the cost neccessary for preparing the input-data for finite element analysis is largely decreased.

1. INTRODUCTION

The finite element method is a useful tool for engineer and scientist, but its result wholly depends on the finite element model being used. Thus, remeshing and its reanalysis are always required for proving the propriety of the finite element solutions. For this purpose a number of mesh generators are already proposed and actually in use.

Most of these automatic mesh generators are written in FORTRAN and work on general-purpose computer. Good finite element model is obtained by the trial-and-error procedure, and the propriety of the model should be judged from the picture of the FE model. Generally

* : Department of Civil Engineering

** : Former Student of Civil Engineering Department

speaking, we don't require fast computation for the automatic mesh generator. These items suggest that micro-computer is the best tool for the proposal of new automatic mesh generator.

In this paper the authors propose a new automatic mesh generator for two-dimensional finite element analysis which works on a micro-computer. The characteristics of the new method are as followings;

(a). Manual is unncessary for the user of the method.

(b). It is useful for generating FEM model for system with very complicated surrounding configuration.

(c). The program is written in BASIC so that it works on micro-computer.

2. BASIC CONCEPT OF AUTOMATIC MESH GENERATION

Automatic mesh generator is required in many engineering fields where the finite element method is used as a tool for their surveying. Thus, there arise many kinds of automatic mesh generators each of which satisfy its own purpose. The mesh generator to be proposed in this paper aims to treat any two-dimensional structural system whose surrounding configuration is complex.

Any complex configuration is easily transformed into a gathering of triangular and quadrilateral shapes by giving appropriate dissection lines into original configuration. It is obvious that the automatic mesh generation for these two types of configurations, i.e. triangular and quadrilateral shapes, is simpler than treating original complex configuration. We introduce this idea in the basis of the new method.

Most of the troubles at the procedure of mesh generation occure at the stages of preparation of input-data, i.e. the geometrical properties of whole system and the informations of how to generate mesh patterns, and giving them into the computer. Before the use of any automatic mesh generator its user is required to prepare all input-data in advance, and as the configuration of the structural system becomes complex, the number of input-data to be prepared neccessarily increases. Furthermore, since most of mesh generators in use are generally very complex for user, its user is neccessarily required to refer to the user's manual during the time of its use. These indicate that the most favourable mesh generator should not require any obligation to the user but it should prepare whole answers beforehand . In other words, the machine should give questions to the user successively in accordance with the procedure of the mesh generating. In this study the machine-oriented mesh generator is newly tried. In the proposed method its user may answer to the questions whole of which are given on the CRT-display, and by answering to them the result is illustrated on the display at the same time. If the user does not satisfy the result, he/ or she can use NENU or EDIT MODE and modify the result in a part or as a whole. The number of input-data is also minimized in the method. The user is required only the coordinates of nodes which are neccessary for the presentation of the configuration of the whole structural system and of the dissection lines to cut it into triangular and/or quadrilateral subareas.

As indicated in the previous section any automatic mesh generator does not require fast computation. Thus, the program of the new method is written in BASIC so that it can be usefully used by the micro-computer.

3. AUTOMATIC MESH GENERATOR

3.1 INPUT-DATA AND OUTPUT

As described in previous section the automatic mesh generator which is proposed in this study is the machine-oriented one. Thus,the number of input-data is minimized as followings;

1.Coordinates of nodes which are neccessary for the presentation of the configuration of total system.

2.Physical properties for subareas.

3.Physical properties for edges of total area and subareas.

Lines connecting nodes and dissection lines cutting whole area into subareas are automatically drawn by answering to questions given by the computer. Physical properties of above items 2 and 3 are prepared for giving , for example, Young's modulus, Poisson's ratio and so on, because subareas are defined for areas with different physical properties.

Output-data of the method are typical ones which are requested at the application of the finite element method. They are as followings;

1.Number of nodes.

2.Number of finite elements.

3.Number of edges.

4.Coordinates of nodes.

5.Element-node incidence.

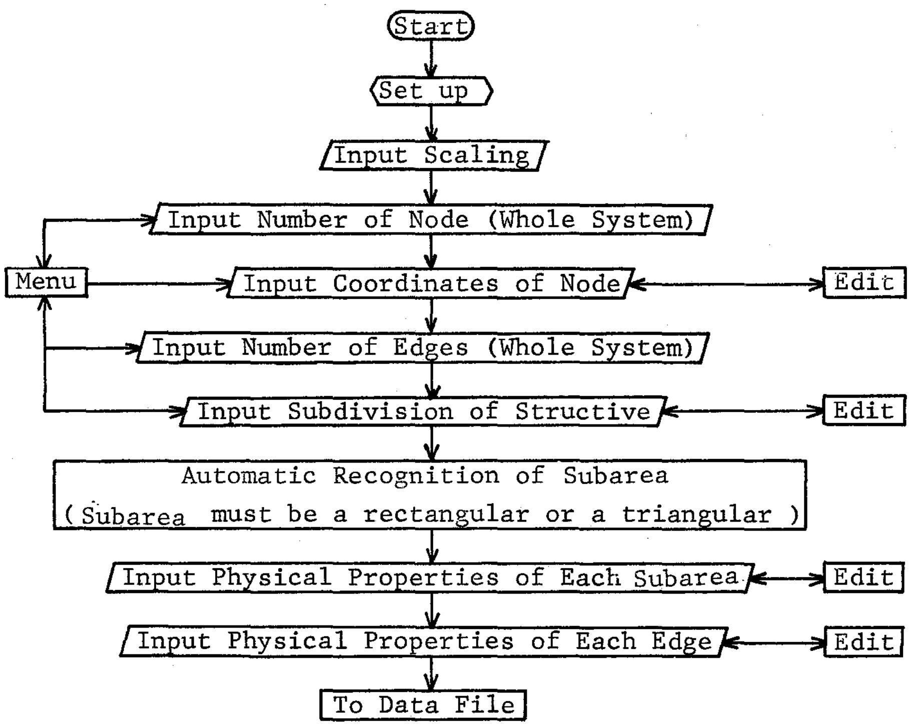
6.Nodes on edges surrounding each subarea.

Items 1 to 5 are typical output data neccessary for the finite element method, and item 6 is an useful information for giving the boundary condition and the applied force vector.

3.2 FLOW-CHART OF AUTOMATIC MESH GENERATOR

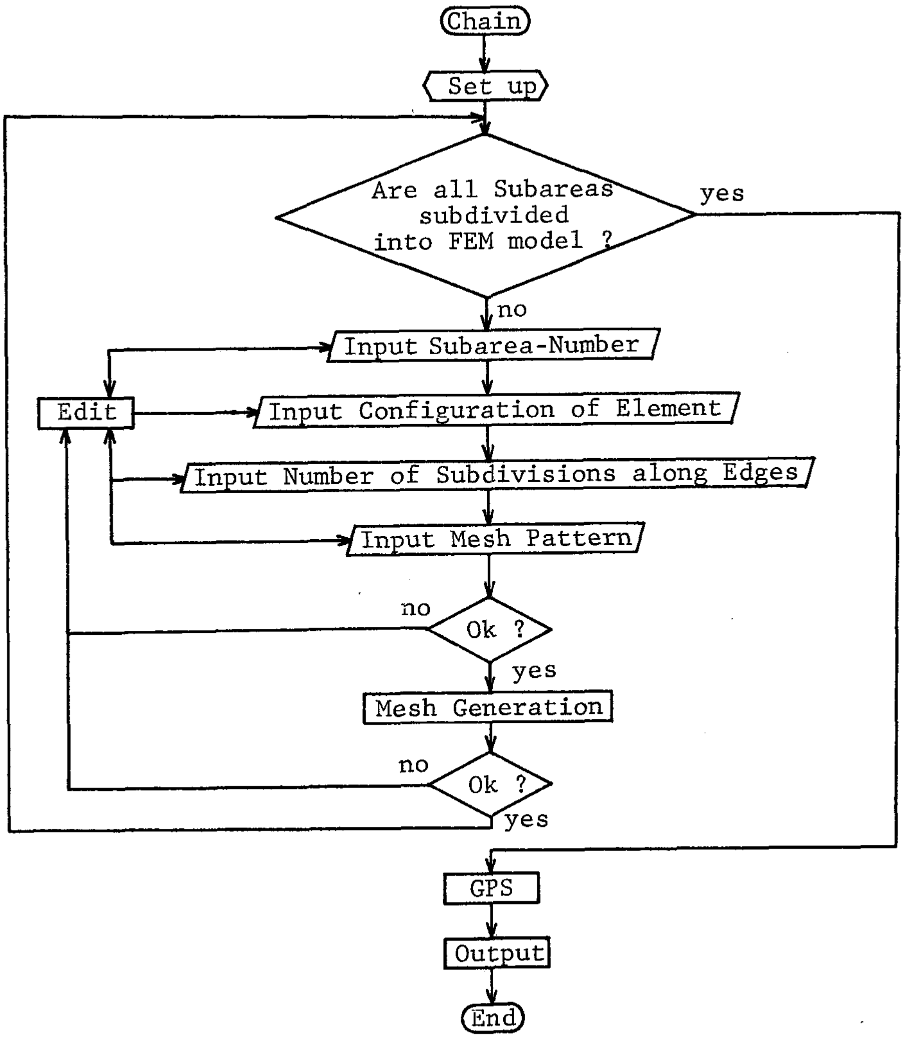
Flow-chart of the proposed method is illustrated in Fig.1. Fundamentally, the method consists of two programs; Subdivision of total system into subareas and Mesh generation of each subarea.

In the first program subareas are generated from the original total structure in accordance with the informations given by the user, and at the end of this program all informations of subareas generated in this program are stored in the file. By the chaining of the second program the generation of the finite element model for each subarea is started. In this program many kind of finite element models are prepared, and in accordance with the selection among them the



(a) FLOW-CHART OF SUBDIVISION OF WHOLE STRUCTURE

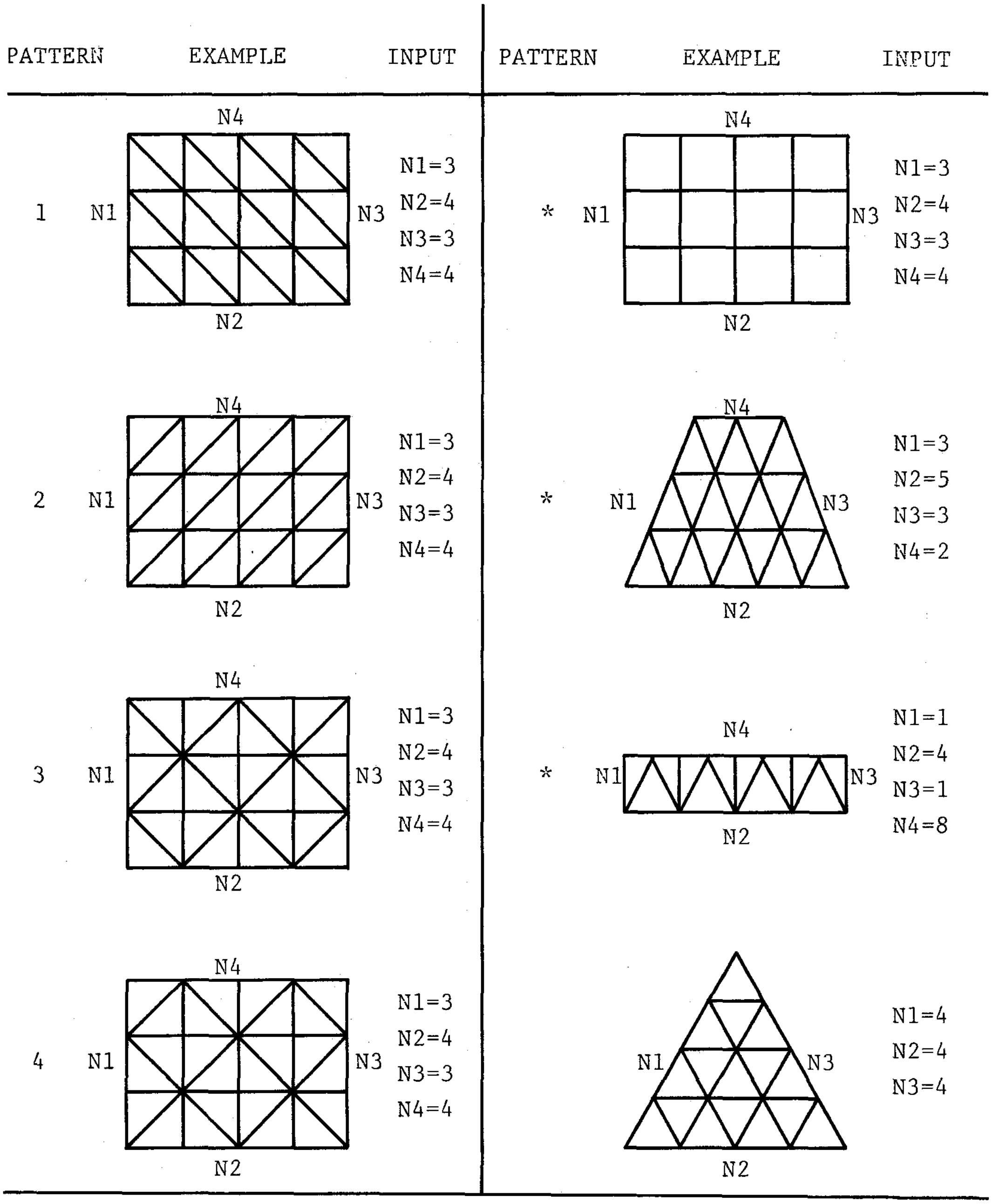
Fig. 1 FLOW-CHART OF PROPOSED MESH GENERATOR



(b) FLOW-CHART OF MESH GENERATION OF EACH SUBAREA

Fig. 1 FLOW-CHART OF PROPOSED MESH GENERATOR

coordinates of all nodes and the incidence between generated nodes and elements are automatically caluculated and the result of the finite element model is illustrated on the display. If the user cann't be satisfied by the result, its modification is , of course, possible. Here, it must be noticed that all of finite element models prepared in this program are generated by using the primary algebra as presented in Fig.2.



Note : PATTERN indicated by (*) is automatically loaded by Previous Step of "Input Configuration of Element".

Fig. 2 MESH PATTERNS AND THEIR TYPICAL EXAMPLES

3.3 DETAILS OF FLOW-CHART

After the start of the first program its user may answer the questions presented on the display.

SUBDIVISION OF TOTAL STRUCTURE

INPUT Scaling ; Give the maximum dimensions of the total structure along x and y axes. By this input the configuration of total system is drawn in a prescribed area of the display.

INPUT Number of nodes ; Give the total number of nodes which are neccessary to draw the configuration of original structural system on the display.

INPUT Coordinates of nodes ; Give the coordinates of nodes of original structural system. In accordance with this input data nodes are generated and illustrated on the display. At the same time node number is successively labelled automatically and it is presented on the display.

INPUT Number of edges ; Give the number of edges which are neccessary to draw the original structural system and also to cut it into subareas.

INPUT Subdivision of structure ; Give the labels of two nodes, then an edge connecting these two nodes is drawn. Repeat this procedure as many times as the number of cutting lines which are neccessary to subdivide the original configuration into a gathering of subareas. "Straight Line" and "Circle Line" are prepared. If the latter is used, give the curvature additionally.

AUTOMATIC RECOGNITION OF SUBAREAS ; If above input data is enough to subdivide the original configuration into a gathering of triangular and/or quadrilateral subareas, then each subarea is automatically recogized and it is labelled and the label is illustrated at the position of the centre of gravity of the subarea. Otherwise, this part does not work. And further input data is required.

INPUT Physical properties of each subarea ; Give physical properties (for example, Young's mudulas, Poissons ratio and so on) of each subarea. Three data spaces are prepared for each subarea. This INPUT is for subareas with different physical properties.

INPUT Physical properties of each edge ; Give physical properties of each edge. Three data spaces are prepared for each edge.

All data presently generated are stored in the file and this program stops.

MESH GENERATION OF EACH SUBAREA

By the chaining of successive program "MESH GENERATION OF EACH SUBAREA" following steps begin to work.

INPUT subarea-number ; Select a subarea and input the label of the area.

ARE ALL SUBAREAS SUBDIVIDED INTO FEM MODEL ? Give "Yes" or "No".

INPUT Configuration of element.

INPUT Number of subdivisions along edges of the subarea presently treated.

INPUT Mesh pattern ; Select a mesh pattern among patterns which are prepared, and input it.

MESH GENERATION ; According to the input data the subarea is subdivided into a finite element model.

OK? ; The result of the finite element model of the subarea is presented on the display. If it is not satisfactory, EDIT MODE works and its modification becomes possible.

GPS ; This step is an optional process of the node reordering method for decreasing the half bandwidth and also the matrix profile. Gibbs-Poole-Stockmeyer algorithm is introduced in this step.

OUTPUT ; Output-data presented in Section 3.1 is obtained. But, note that the node labels are the ones newly obtained by GPS.

4. EXAMPLES

In this section we present an example of mesh generation by proposed method. A series of pictures of the display according to the procedure are illustrated in Fig.3.

5. CONCLUDING REMARKS

In this paper we showed a new automatic mesh generator which works on a micro-computer, and an example of finite element model generated by the method was illustrated. At using proposed method the user is not required to refer the manual. By answering to the questions successively presented on the display a finite element model is obtained for each subarea and consequently we obtain the whole finite element model.

<部分系節点座標 の 入力>

部分系節点数 : 23

```
節点     X座標    ,    Y座標
 11  .108787E+02 .121213E+02
 12  .100000E+02 .100000E+02
 13  .108787E+02 .787870E+01
 14  .130000E+02 .170000E+02
 15  .130000E+02 .130000E+02
 16  .130000E+02 .700000E+01
 17  .130000E+02 .300000E+01
 18  .151213E+02 .121213E+02
 19  .160000E+02 .100000E+02
 20 ? 15.1213,7.8787
```

［f・9］修正

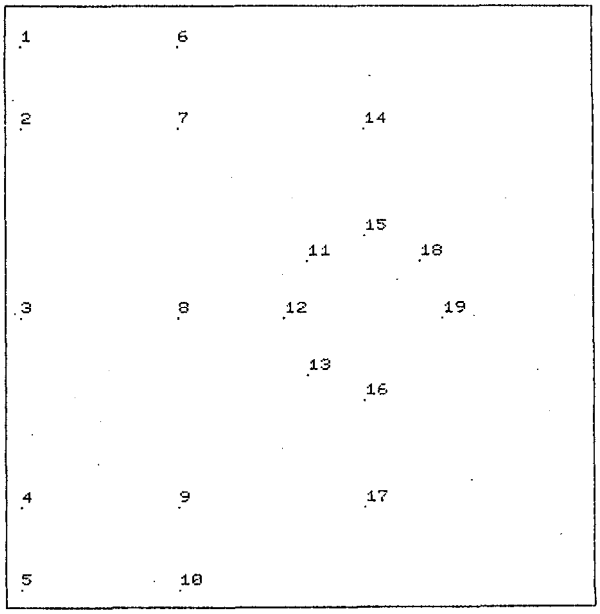


Fig. 3 AN EXAMPLE OF AUTOMATIC MESH GENERATION
STAGE 1 ; INPUT OF NODES

<部分系辺－節点関係 の 入力>

部分系 辺数 : 36

```
節点   辺   節点  L/C   半径
 11 ==21== 15    1 .300000E+01
 11 ==22== 12    1 .300000E+01
 12 ==23== 13    1 .300000E+01
 13 ==24== 16    1 .300000E+01
 16 ==25== 17    0 .000000E+00
 14 ==26== 21    1 .700000E+01
 15 ==27== 18    1 .300000E+01
 18 ==28== 19    1 .300000E+01
 19 ==29== 20    1 .300000E+01
 16 ==30== 20  ?
```

［f・9］修正

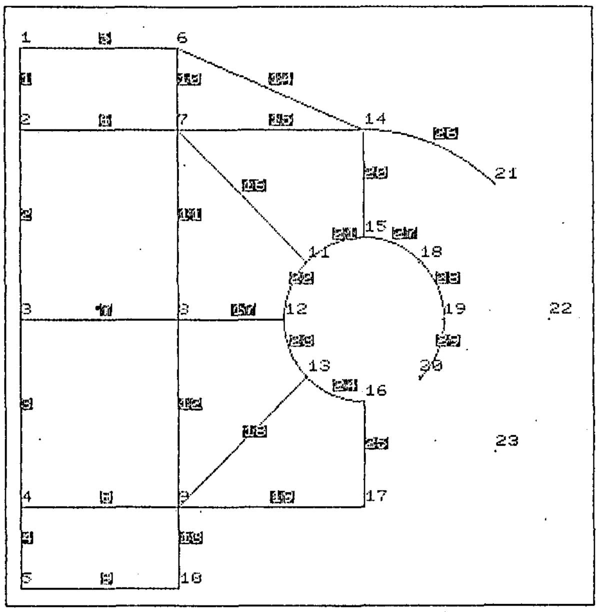


Fig. 3 AN EXAMPLE OF AUTOMATIC MESH GENERATION
STAGE 2 ; INPUT OF LINES

```
<部分形物理定数の入力>

部分形数  : 13
       形
No      1           2           3
 1+.100000E+01+.100000E+03-.100000E+01
 2+.200000E+01+.200000E+03-.200000E+01
 3+.300000E+01+.300000E+03-.300000E+01
 4+.400000E+01+.400000E+03-.400000E+01
 5+.500000E+01+.500000E+03-.500000E+01
 6+.600000E+01+.600000E+03-.600000E+01
 7+.700000E+01+.700000E+03-.700000E+01
 8+.800000E+01+.800000E+03-.800000E+01
 9+.900000E+01+.900000E+03-.900000E+01
10 ? 10       ? 1000     ? -10

[f・9] 修正
```

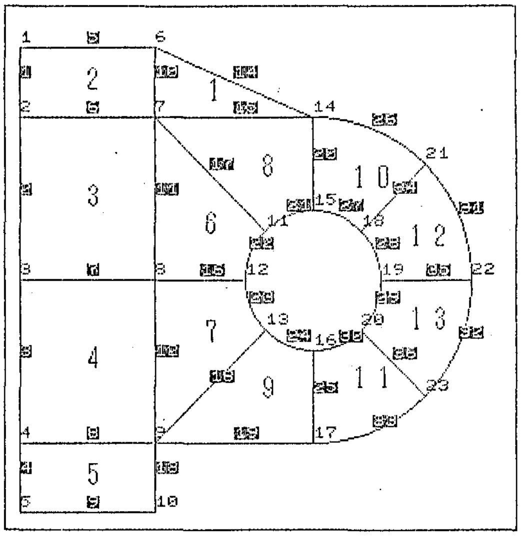


Fig. 3 AN EXAMPLE OF AUTOMATIC MESH GENERATION
STAGE 3 ; COMPLETION OF TOTAL SYSTEM

```
<部分形の分割>

部分形の番号（修了：99）: 10
  部分形の数  : 13

要素の種類（3  or  4）:  3

辺の分割数
  辺番号    20          :  4
  辺番号    27          :  3
  辺番号    34          :  4
  辺番号    26          :  3

分割のパターン          :  3

分割はOKですか

 OK ? ( Yes : 1  or  No : 0 )
```

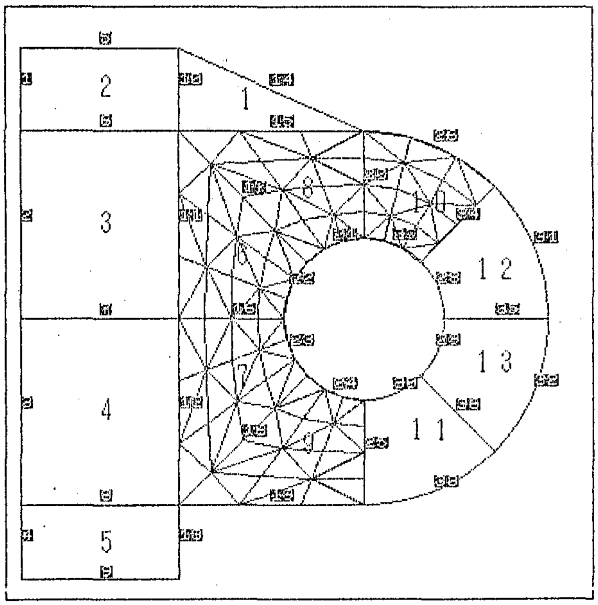


Fig. 3 AN EXAMPLE OF AUTOMATIC MESH GENERATION
STAGE 4 ; MESH GENERATION OF SUBAREA 10

＜部分形の分割＞

部分形の番号（修了：９９）：　３
部分形の数　：13

要素の種類（３　ｏｒ　４）：　３

辺の分割数
辺番号　2　：　3
辺番号　7　：　5
辺番号　11　：　3
辺番号　6　：　5

分割のパターン　：　１

分割はＯＫですか

OK ? (Yes : 1 or No : 0) 1

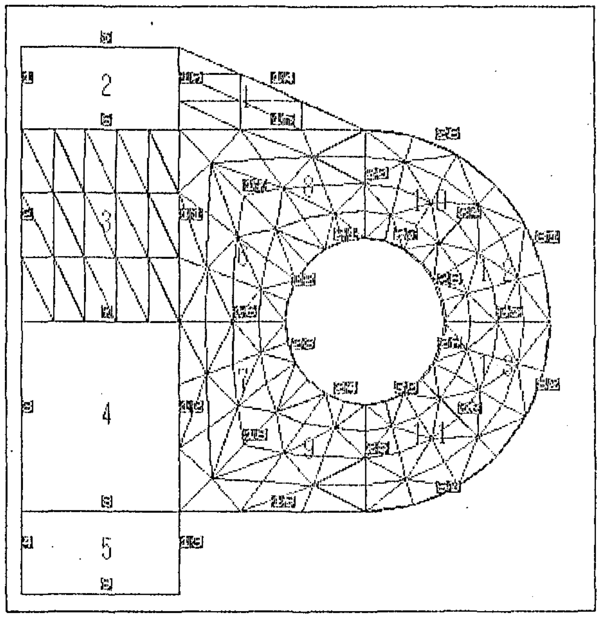


Fig. 3 AN EXAMPLE OF AUTOMATIC MESH GENERATION
STAGE 5 ; MESH GENERATION OF SUBAREA 3

＜部分形の分割＞

部分形の番号（修了：９９）：　５
部分形の数　：13

要素の種類（３　ｏｒ　４）：　３

辺の分割数
辺番号　4　：　2
辺番号　9　：　5
辺番号　13　：　2
辺番号　8　：　5

分割のパターン　：　２

分割はＯＫですか

OK ? (Yes : 1 or No : 0) 1

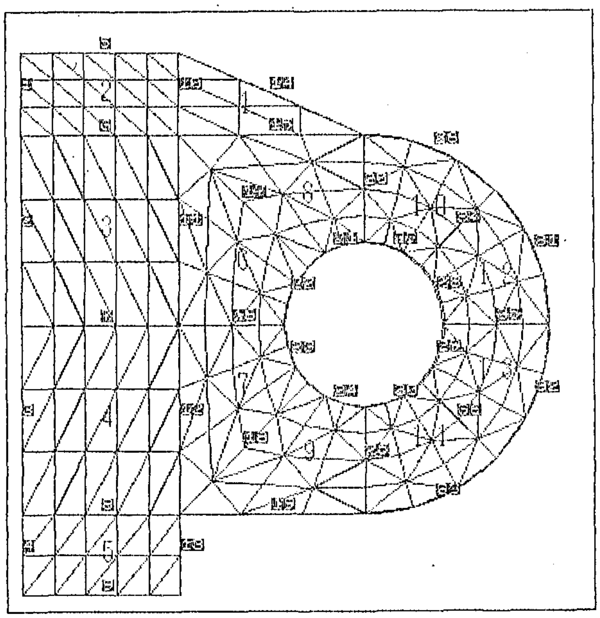


Fig. 3 AN EXAMPLE OF AUTOMATIC MESH GENERATION
STAGE 6 ; COMPLETION OF FE MODEL

As shown in the example the proposed method is effective, but since all of the automatic mesh generators presently prepared for the dissection of each subarea are based on the primary algebra, they lack the flexibility. That is, if the number of nodes on one edge is given, it restricts the dissections of subareas neighbouring to the edge. Especially, if whole area is subdivided by using only triangular subarea, the input of the number of disection of one edge determines the disection of whole area.

In order to increase the flexibility of the disection method a new subdivision method for triangular and quadrilateral area which is not based on the algebra must be prepared and added in this program.

Present program of the proposed method is developed by using PC9801 vm2 of NEC.

ACKNOWLEDGEMENT

The authors acknowledge and thank Mr. K. Sakamoto, former student of Structural Engineering Laboratory, Department of Civil Engineering, Okayama University, for the assistance of developing this complicated program.

REFERENCES

1). T.Taniguchi and E. Kajitani,"Automatic mesh generator for finite element method",Symposium of Chuhgoku-Shikoku Area, (JSCE),1984, pp.31-32

2). T. Taniguchi, K. Sakamoto and T.M. Ikarimoto,"Automatic mesh generator for crack propagation analysis",Symposium of Chuhgoku-Shikoku Area, (JSCE), 1985, pp.33-34