Finite Element Mesh Generation in the Framework of an Expert System
(Pre-Post Program "SCOPE" – Interface to MSC/NASTRAN)

by

Koichi Mikami

Yoshio Iwahashi

Osamu Murakawa

Tsuneo Sakato

Sumitomo Heavy Industries, LTD.

ABSTRACT

This paper describes an expert system to automatically generate finite elements which is a subsystem of finite element pre-processing system named "SCOPE".

In SCOPE, the geometric shape of analysis model is represented as a collection of bicubic parametric surface patches and finite elements are generated through the mapping of mesh divisions in the parametric space. The task of selecting the number of division for each surface patch is carried out by an expert system. The problem is formulated as multiobjective optimization with respect to the total number of elements, element aspect ratio and element size. The actual optimization process is implemented by using the concept of the Generalized Problem Solving as applied to a robot planning.

The effectiveness of the approach is demonstrated by several examples and the potential expansion of the method is discussed.

1. INTRODUCTION

With the advancement of the computer technology, highly accurate finite element analysis using complex and large size models had become practical. However, much more time of the engineer is required for pre-post processing for input data preparation and results evaluation. Many pre-post processors have been developed to solve this problem and have been used in the engineering analysis. To improve the user interface to finite element analysis softwares such as MSC/NASTRAN, we recently developed finite element analysis pre-post program "SCOPE" (Structural Concept Oriented Program for Engineers) which has many characteristics as follows:

- interactive color graphics
- command selection by the menu
- geometric modeling
- MSC/NASTRAN input-output interface

At the same time, an effort has been made to develop an expert system which automatically generates finite element meshes.

The first expert system applied to structural analysis is SACON (Structural Analysis CONsultant) [1], which advises analysts in the use of finite element program MARC. Since then, many programs have been developed to assist engineers in the complicated process of finite element analysis. For examples, the prototype expert system, called FEMOD (Finite Element MODeling) [2], gives the designer advice in modeling for a special finite element program EAL (Engineering Analysis Language). Recently, the system called ADEPT (Automated DEsign exPerT) [3] was reported. This system recommends the finite element analysis strategy using informations given by the user and "textbook" level knowledge. These systems can be classified into "interpretation" or "diagnosis" type.

The task of mesh generation in the finite element analysis, on the other hand, is of a planning type. It corresponds to the selection of basis for series expansion in the boundary value problems of partial differential equations, for which insights of the analyst is essential and some experience and skill are indispensable. In the present paper, we combined goal programming in the multiobjective optimization theory [4] with the concept of Generalized Problem Solving as applied to a robot planning [5]. The effectiveness of this approach for mesh generation is shown by several examples.

2. FINITE ELEMENT MODELING

> *The beginning of the Problem Formulation Phase occurs in the mind of the analyst. He contemplates nature (or his navel, or whatever), decides what he needs to know, and constructs a mathematical problem whose solution, he hopes, will provide relevant answers to his questions. He will, naturally, require computational tools to solve his mathematical problem and, fortunately or unfortunately, the available tools have a strong influence on the analyst's choice of a mathematical problem. It would, after all, do no good to formulate a problem that could not be solved.* Section 3.4.1 Structural Modeling, p.3.4-1 by McCormick from [6].

2.1 Task Analysis

The process of finite element modeling in "SCOPE" is shown in Fig.2.1. First, idealizing the analysis object, the geometric model is generated and its material and element properties are defined. Then, the geometric model is devided into a number of finite elements and grid points for the analysis degree of freedom are generated. After that, loads and constraints to define the analysis case are applied to those grid points and/or elements. Finally, all the data are output in the format appropriate to the analysis by MSC/NASTRAN.

All of these tasks of finite element modeling can be considered as promising fields of expert system application, where substantial benefit may be obtained if the knowledge of expert is used in the system development.

Particularly, finite element generation or the mesh division of the geometric model is one of the most effective tasks. It requires tradeoff between accuracy and cost, the result of which depends on the analyst's skill, experience and knowledge. Also, it is generally the most time consuming part of the process.

2.2 Geometric Model and Mesh Division

Cubic polynomial representation of the geometric model is adopted in "SCOPE". Namely, the shape of the analysis model is represented as a collection of cubic parametric curves, bicubic surface patches and tricubic solids. A detailed description of surface patches, shown in Fig.2.2 will given in the following.

The surface patch is defined by a cubic equations of two parameters s and t, and can be written as follows:

$$P(s,\ t) = SAT^{T},\tag{2.1}$$

where $P(s,\ t)$ is a vector of which the components are x, y and z, $S=[\ s^3\ \ s^2\ \ s\ \ 1]$, $T=[\ t^3\ \ t^2\ \ t\ \ 1]$, $0 \leqq s$ and $t \leqq 1$, and T^{T} is the transposed matrix of T. And A is as follows:

$$A=\begin{bmatrix} a_{00} & a_{01} & a_{02} & a_{03} \\ a_{10} & a_{11} & a_{12} & a_{13} \\ a_{20} & a_{21} & a_{22} & a_{23} \\ a_{30} & a_{31} & a_{32} & a_{33} \end{bmatrix},\tag{2.2}$$

which represents coefficients of bicubic polynomials and is called Algebraic Coefficients (AC). These algebraic coefficients can be transformed to Geometric Coefficients (GC) using Hermite or Bezier form. Varying two parameters s and t from 0 to 1 gives all points on a surface patch.

Finite elements are generated through the mapping of mesh divisions in the parametric space. When the number of division along s- and t- directions of a patch, n_s and n_t , are given, "SCOPE" automatically generates points on the surface patch by interpolation and generates $n_s \times n_t$ quadrilateral finite elements. For example, suppose the rectangular surface patch as shown in Fig.2.3 and if $n_s = 4$ and $n_t = 3$, finite elements are generated as shown in Fig.2.4.

2.3 Simplifying Assumptions and Edge Class

In order to establish the framework of the program, following simplifying assumptions are introduced.

1) the analysis model consists of plate or shell type structure, the geometry of which can be represented by its neutral surfaces.
2) the geometry of the analysis model is "completely" decomposed into four sided patches in a sense that any two continuous patches are connected only through their common edge and corresponding common vertices.
3) only the regular division is allowed as the method of mesh division where each patch is equidistantly divided in the parameter space in the s- and t- directions. Accordingly, opposite edges of a patch have the same number of division.

These assumptions seems to restrict the range of applicability of the program. However, it is still useful and these restrictions may be removed once the main framework of the program has established.

From the above assumptions 2) and 3), it follows that there exist subsets of edges, each member of which have a common number of mesh division. We call these subsets as "edge class".

An example of classification of edges (edge class) is shown in Fig.3.1(a). The model is for a rectangular thin plate with a circular hole subjected to a uniform tension. Its cognitive matrix is shown in Fig.3.1(b). As seen from these figures, a patch can be represented by two edge classes intersecting with each other.

Assigning the number of division to a edge class means assigning one number to all edges belonging to this edge class and it is sufficient to assign a set of integers to edge classes.

3. FRAMEWORK OF THE EXPERT SYSTEM

It is now clear enough that the problem is to find a set of integers representing numbers of subdivision of surface patches which can be considered to be "acceptable" for the analysis. Although the problem can be formulated as goal programming in the mathematical theory of multiobjective optimization as shown in the following, the actual optimization process is implemented by using the concept of Generalized Problem Solving applied to a robot planning. These two methods of planning have essential similarity and can conveniently be combined when the numerical treatment of the problem is possible. The concept of Generalized Problem Solving gives a flexible and general framework to simulate the analyst, whereas the goal programming approach extremely simplify the search algorithm when there exist interactions among objectives or subgoals.

3.1 Goal Programming

It is one of the most difficult question to answer what is the "acceptable" mesh division. It largely depends on the analyst's knowledge, experience, skill, preference, etc. As the measure of the quality of the mesh division, we adopted here following three parameters:

1) element aspect ratio
2) element size
3) the total number of elements

and currently assumed their ideal values as follows:

1) element aspect ratio is 1.0
2) element size is uniform (including some weighting parameter for exceptional edges)
3) the total number of elements is to be given by the user

Although the total number of elements is already the result of some tradeoff between the analysis cost and accuracy, we assumed it to be given by the user for the sake of simplicity. Obviously, only these three parameters are not sufficient and any factor affecting the mesh quality may be included.

As a matter of fact, it is not possible to bring these parameters to their ideal values at the same time and some tradeoff is required. This problem can be formulated using goal programming in the multiobjective optimization theory, which can be written in a general form as

$$\text{minimize } \| f(x)-f^{*} \|_{q} \quad \text{subject to } x \in \mathbf{X}, \tag{3.1}$$

where $f(x)$ is the vector objective function of the decision variable x, f^{*} is the goal or ideal point in the objective space to be selected by a decision maker, $\mathbf{X}$ is the feasible region, and the condition $x \in \mathbf{X}$ represents the constraint. $\| \cdot \|_{q}$ stands for a norm in the objective space and may be expressed as

$$\| f(x)-f^{*} \|_{q} = [\sum_{k=1}^{p} | w_{k} (f_{k}(x)-f_{k}^{*})|^{q}]^{1/q}, \quad q \geqq 1 \tag{3.2}$$

where $f_{k}(x)$ is the component of $f(x)$, f_{k}^{*} is the ideal value for each objective function, w_{k} are weighting parameters and p and q are integers. This function is often called as the regret function.

Decision variables are numbers of division along s- and t- edge of each patch, i.e. numbers of division of edge classes. Other dependent variables expressing the state of mesh division is decided by a set of these numbers. In the present system, we adopted the following functional form for the regret function to evaluate the quality of the mesh division.

$$\sum_{k=1}^{3} | w_k (f_k - f_k^*) |, \qquad (3.3)$$

$$f_1 = \max[x_1], \qquad f_1^* = 1.0,$$

$$f_2 = \max[x_2]/\min[x_2], \qquad f_2^* = 1.0,$$

$$f_3 = x_3 / N_r, \qquad f_3^* = 1.0,$$

N_r : total number of elements given by the user

where x_1 , x_2 and x_3 are element aspect ratio, edge length and the total number of elements respectively; max[x] and min[x] mean the maximum and minimum values of x; values of weighting parameters w_1 , w_2 and w_3 are set to 0.5, 0.25 and 2.0 respectively. This means that when maximum element aspect ratio is 3.0, maximum to minimum length of edges is 5.0 and the total number of elements is $2N_r$, the value of each objective function is 1.0. These objective functions are shown in Fig.3.2(a), (b) and (c), respectively.

3.2 Optimization process

Since it is not practical to get explicit functional dependency of Eq.(3.1), we proceed to implement an actual optimizing process in terms of the concept of Generalized Problem Solving. There are three components characteristic to the Generalized Problem Solving, a set of states, a set of actions or operators that transform one state into another and goal (state) which satisfies specified constraints. Starting from an arbitrarily given initial state, a problem solver applies a sequence of actions in attempt to reach the desired goal (state).

For the present problem of mesh generation, state representation primarily consists of a set of integers assigned to the edge classes and actions correspond to the change of those integers. Table 3.1 shows the data structure representing the state of mesh division which consists of geometric objects such as points, edges, surface patches, edge classes and their attributes. Relations among points, edges and surface patches have a hierarchical structure. If the number of division of each edge class are given, values of each attribute of geometric objects can be calculated.

Fig.3.3 shows the optimization process used in the present system. First, the initial mesh is arbitrarily selected and becomes the first current mesh. And then, candidate (actions) to transform the current mesh into another are generated, corresponding states of mesh division are evaluated by the regret function and only one of the best actions is selected. Optimization is performed by the application of a series of these actions to reach from the initial mesh to the ideal mesh. Finally, optimaized finite elements is output as the number of division along s- and t- edge for each patch.

Initial state

The initial number of division of edges (all edge classes) n is given as follows:

$$n = \mathrm{ANINT}[\sqrt{(N_r / N_p)}], \qquad (3.4)$$

N_r : the total number of finite elements input by user

N_p : the number of patches in a geometric model.

ANINT[x] : the largest integer that does not exceed the magnitude of x+0.5 for a real and positive number x

This means that each patch have initially the same number of elements. Therefore, the total number of finite elements for this state is nearly equal to N_r and aspect ratio and mesh size are proportional to these of initially given surface patches. However, there will be some elements of large aspect ratio by which calculation errors will be caused and of much great size difference between elements. In such a case, the initial mesh can be improved by applying following actions.

Updating actions

To improve current mesh, following three strategies are considered.

(1) improvement of maximum element aspect ratio

(2) decreasing the difference of length between elements edges.

(3) adjustment of the current total number of elements to the specified number

For these purposes, following concrete alternative actions are selected:

(1) increasing the number of division by 1 for longer edge of a patch of maximum element aspect ratio

(2) decreasing the number of division by 1 for shorter edge of patches of maximum element aspect ratio

(3) increasing the number of division by 1 for the edge of maximum length

(4) decreasing the number of division by 1 for the edge of minimum length

optimization process and stop condition

The value for each action is calculated by the regret function Eq.(3.2) and only one action of minimum value is selected. If the value of this action is less than the current value, the number of mesh division in the data structure is changed and all the dependent variables are replaced. The expert system makes a series of actions as repeatedly as possible.

If all the values for actions are greater than the current value, namely when the value of the regret function became minimum, the expert system stops updating actions and optimized finite element meshes are output as the final number of division along s- and t- edge for each patch.

4. EXAMPLES

Three examples are presented to demonstrate the effectiveness of the expert system's approach . First example is a thin plate with a circular hole as shown in section 2.3. The specified total number of finite elements N_r is 50. The initial state of mesh division is shown in Fig.4.1(b). The number of division for each edge is 4. Thus, the number of elements for each patch is 16 and the total number of elements is 48. The final state of mesh division is shown in Fig.4.1(c), where the total number of elements is 56. From this result, it is found that only element aspect ratios of the largest aspect ratio patch are improved.

Second example is CFRP (Carbon Fiber Reinforced Plastic) strut designed to carry tensile and compressive loads. A quarter geometry model decomposed into 13 patches is shown in Fig.4.2(a). The specified total number of finite elements N_r is 100. The initial state of mesh division is shown in Fig.4.2(b). The number of division for each edge is 3. Thus, the number of elements for each patch is 3 and the total number of elements is 117. The final state of mesh division is shown in Fig.4.2(c), where the total number of elements is 100. Largest element aspect ratios and the difference of edge length are improved.

Final example is a natural vibration analysis model of a gear box, which is represented by 48 patches as shown in Fig.4.3(a). The desired total number of finite elements N_r is 200. The initial state of mesh division is shown in Fig.4.3(b). The number of division for each edge is 2. Thus, the number of elements for each patch is 4 and the total number of elements is 192. The resultant state of mesh division is shown in Fig.4.3(c), where the number of elements is 200. Similary to above two examples, largest element aspect ratios and the difference of edge length are improved.

From these results, the present system give satisfactory results and enables the user to quickly generate the finite element meshes.

5. FUTURE SCOPE

The expert system, described so far, will be expanded by improving following aspects.

1) introduction of graded mesh division method allowing localized fine mesh regions of stress concentration
2) utilization of the information on the loads and constraints conditions
3) inclusion of beam and solid elements

It should be noted further that the framework of the expert system is of a planning type and range of its applicability is not restricted to the finite element mesh generation. For example, design expert systems concerned with decision of optimum design variables will be one of the most promising fields of application of the present approach.

6. CONCLUSIONS

We developed an expert system for the mesh division task in finite element modeling using the goal programming formulae in the multiobjective optimization theory and the framework of Generalized Problem Solving as applied to robot planning. It is found through three application examples that this approach is very effective for the mesh generation and the present system can automatically generate the finite element meshes from given geometric objects.

REFERENCES

1. Bennet J.S.,Engelmore R.S.,"SACON: A Knowledge-based Consultative for Structural Analysis ," Proc. of the IJCAI,Vol.6,1979,pp.47-49.
2. Chen J.L.,Hajela P.,"FEMOD: A CONSULTATIVE EXPERT SYSTEM FOR FINITE ELEMENT MODELING," Conputer & Structure, Vol.29, No.1, 1988,pp.99-109.
3. Robert H.Holt, "ADEPT : An Expert System for Finite Element Modeling," IEEE Proc. Int. Conf. Cybern.Soc.,Vol.2,1986,pp.1462-1467.
4. For example, Keenly,R.L., Decision with Multiple Objects, Preference and Value Trade-off, John Wiley, 1976.
5. Fikes, R.E., Nilson,N.J. and Hart,P.E.,"Learning and Excuting Generalized Robot Plans," Artificial Inteligence , Vol.3, 1972, pp.251-288.
6. The Nastran Theoretical Manual , MSR-40, The MacNeal-Schwendler Corporation, Los Angeles, California, 1972, p.3.4-1.
7. J.D.Foley and A.Van Dam, Fundamentals of Interactive Computer Graphics, Addison-Wesley Publishing Company, Inc., 1982.

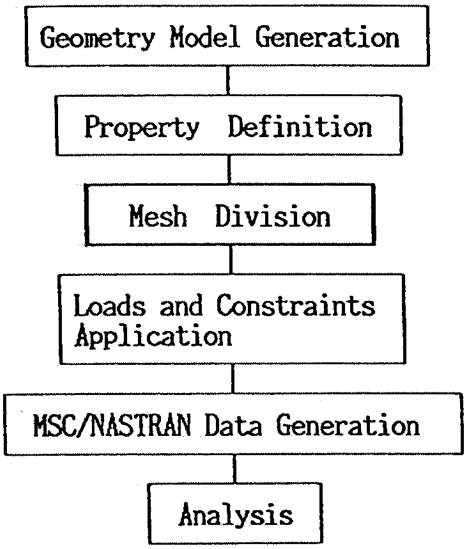


Fig.2.1 Tasks for finite element analysis

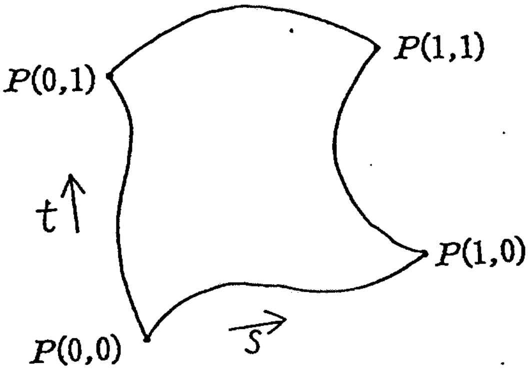


Fig.2.2 A surface patch geometry

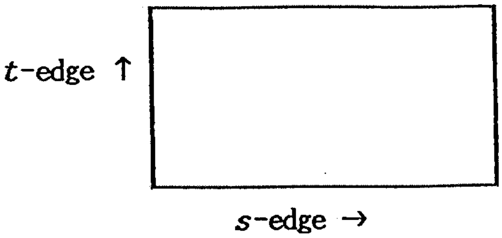


Fig.2.3 Surface patch

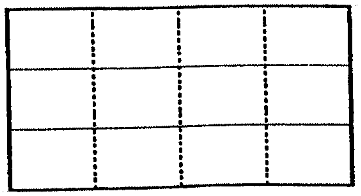
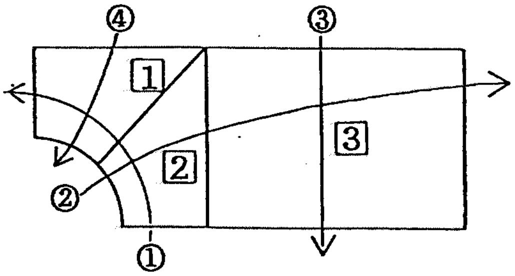


Fig.2.4 Finite element meshes



(a) A plate with a circuler hole (a qurater model)

CLASS No.	①	②	③	④
①	①	$\boxed{2}$		$\boxed{1}$
②	$\boxed{2}$	②	$\boxed{3}$	
③		$\boxed{3}$	③	
④	$\boxed{1}$			④

ⓘ ; CLASS
$\boxed{i}$; patch

(b) Cognitive matrix

Fig.3.1 Example of classification of edges

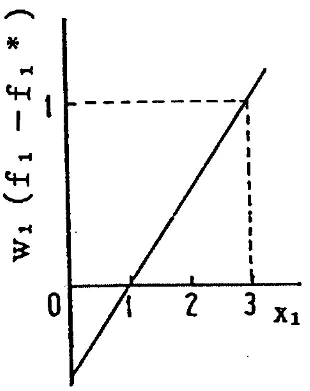


Fig.3.2 (a)

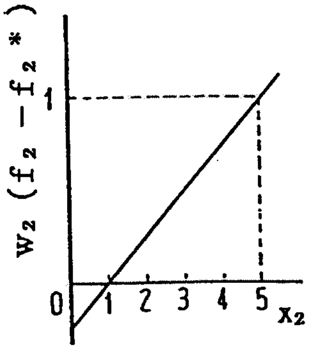


Fig.3.2 (b)

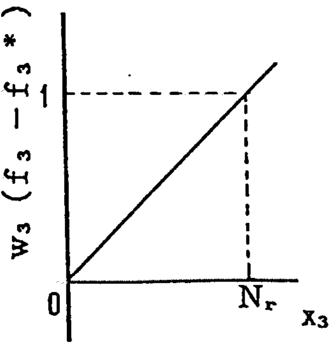


Fig.3.2 (c)

Table 3.1 Data structure

object name	attributes
PATCH	id current element aspect ratio changed element aspect ratio current number of elements changed number of elements *s*-edge id *t*-edge id
EDGE	id class id current length changed length start point id end point id
POINT	id x coordinate y coordinate z coordinate
CLASS	id current number of division changed number of division

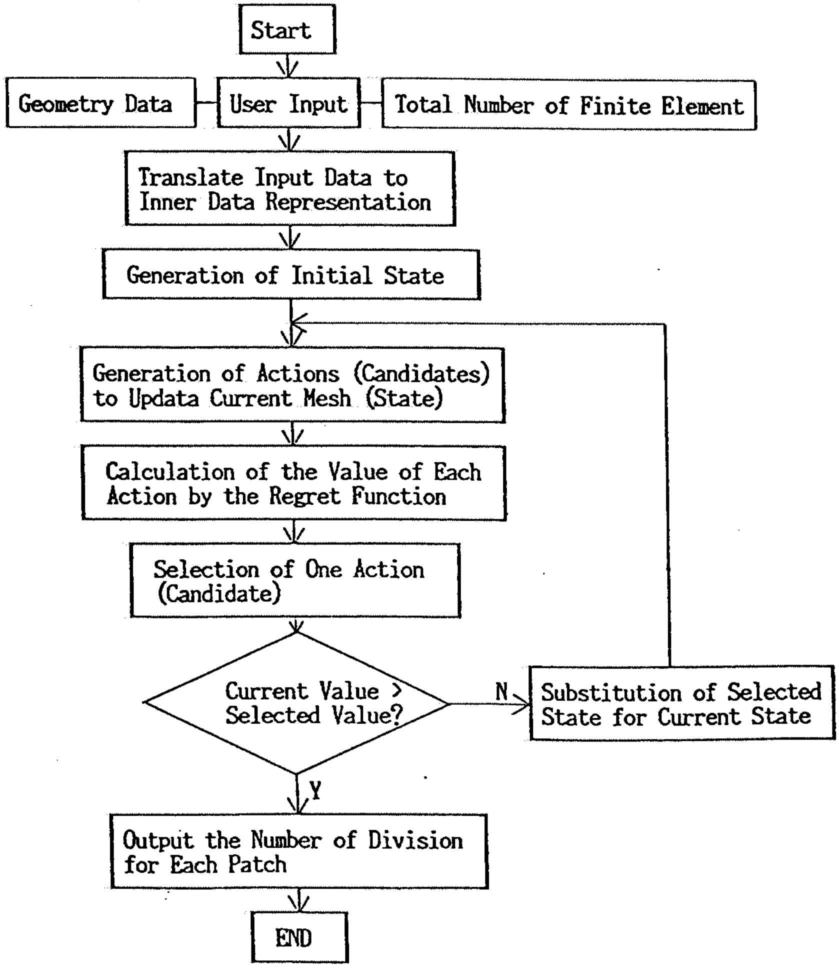
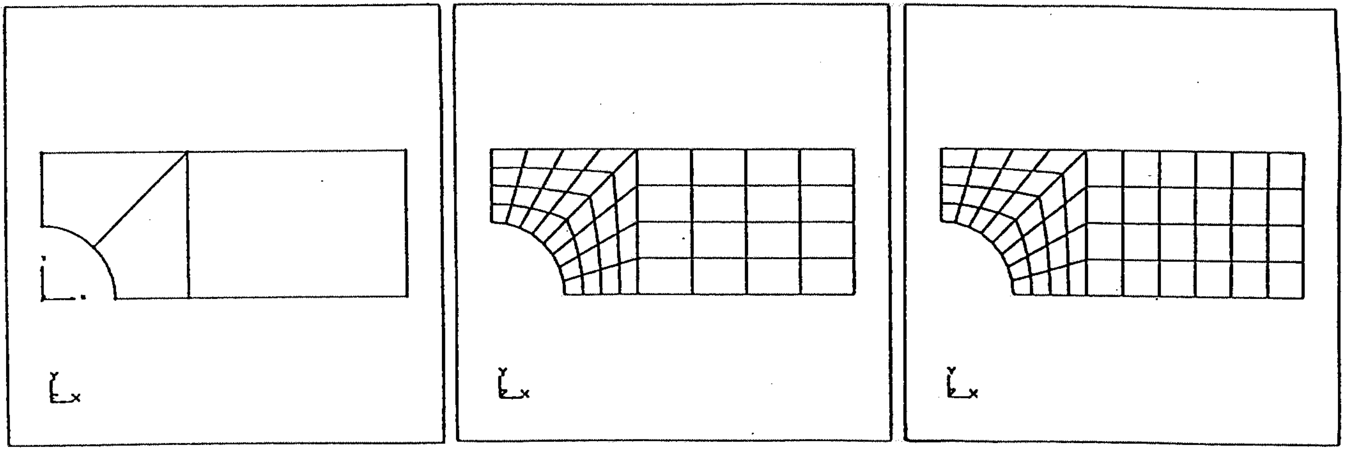
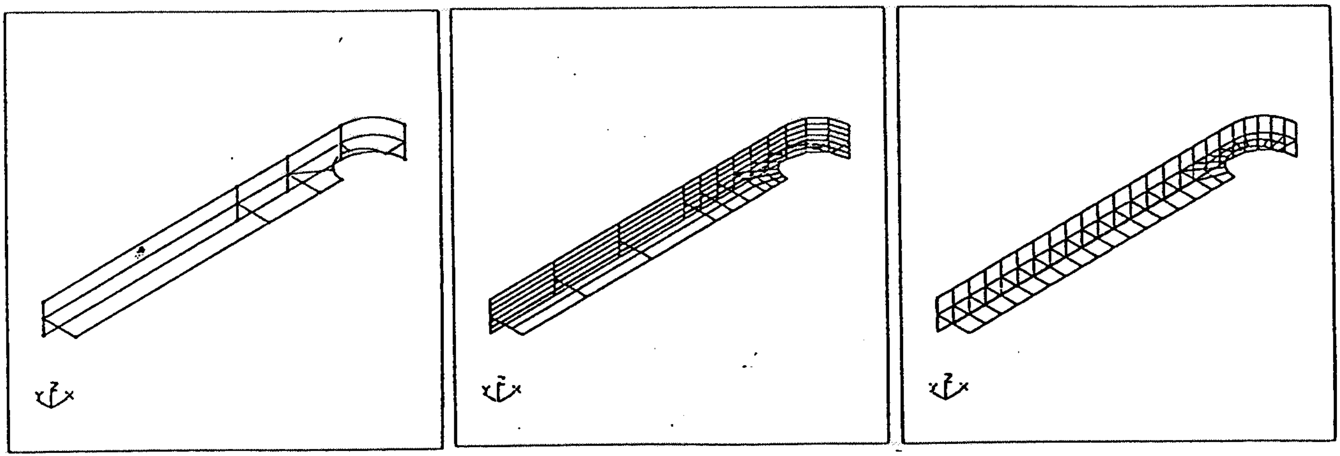


Fig.3.3 Optimization process of the expert system



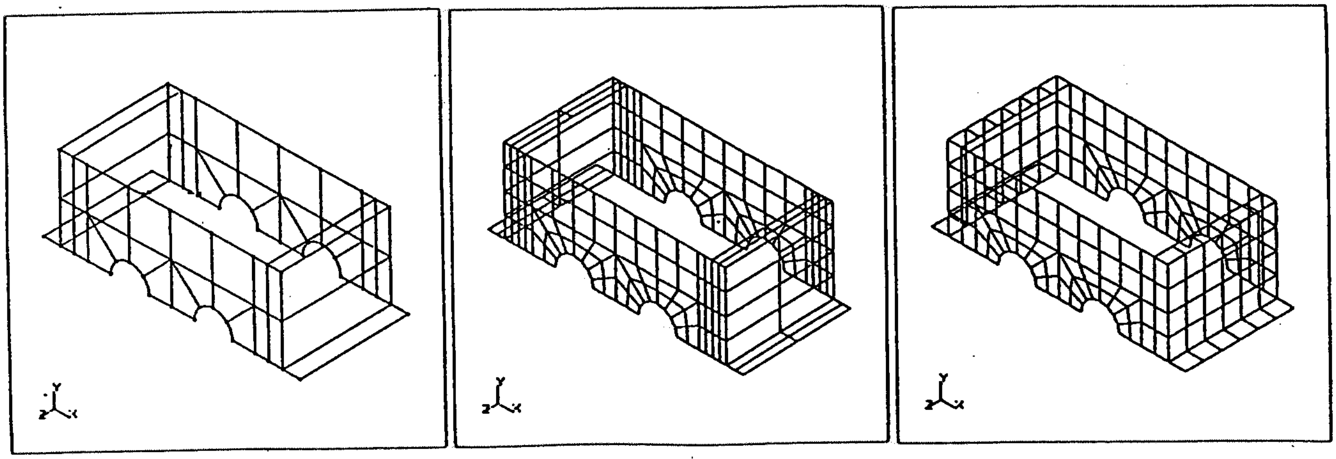
(a) Geometry model (b) Initial mesh (c) Final mesh

Fig.4.1 Example 1 (Plate with a hole)



(a) Geometry model (b) Initial mesh (c) Final mesh

Fig.4.2 Example 2 (CFRP strut)



(a) Geometry model (b) Initial mesh (c) Final mesh

Fig.4.3 Example 3 (Gear box)