CONTROLLED NON-CONFORMING FINITE ELEMENTS AND DATA BASE AS APPROACH TO THE ANALYSIS OF AIRCRAFT STRUCTURE

Z. BOJANIĆ and M. JOSIFOVIĆ
Faculty of Mechanicall eng.
Belgrade, Yugoslavia

Abstract

In this paper a modification of finite element methode is suggested which, at relatively small number of elements gives better solution of nodal displacement.

The modification is based on analysis of convergence condition of finite element methode, conditions which are to satisfy shape function of conforming elements, as well as on the base of behaviour of non-conforming elements analysis, which being applied usualy give a quick convergence solution.

The leading idea of modification of finite element methode consist of using a quick convergence of non-conforming elements and monotonic convergence of conforming elements. A "controlled non-conforming element" is introduced for that purpose which is assigned to fundamental conforming element and its shape function is addopted in this form.

$$\bar{N}_i(\bar{x}) = (1 - \alpha)N_i^k(\bar{x}) + \alpha N_i^r(\bar{x})$$

where $N_i^k(\bar{x})$ - is a clasical shape function which corresponds to fundamental conforming element,

$N_i^r(\bar{x})$ - additional, corrective shape function which describes the shape of equilibated displacemen within an element.

Functions $N_i^r(\bar{x})$ represent those displacements within an element which are effect of nodal forces K_{ij}. These functions do not fulfil continuity conditions on inter-element boundaries, so they correspond to non-conforming elements and are defined numerically. Well known patch test id used for defining parametar α value for group of elements, on the data base regarding simillar construction behaviour. Parametar value depends on relative diameter of finite element and its ratio is between 0 and 1. For $\alpha = 0$ conforming fundamental element is obtained while for $\alpha = 1$ a completely non - conforming, equilibrated element is obtained. Solutions limitting exact solution from upper and lowere boundaries correspond to these elements, while parametar α defines the exact solution.

On the base of a defined parametar α, group of elements, as sub-structure has stiffness matrix which in the best way approximates the real local stiffness of the part of the real structure. Forming the data base based on previous conciderations of the parts of the structure, a base for quick and more real analysis of aeronautical construction is obtained. Having in mind that the structure is modeled with smaller number of elements, that is smaller number of global nodes this methode makes possible very quick analysis which at the same time are exact.

Introduction

Conforming elements that is, elements fulifiling continuity of displacements conditions at the inter-element boundaries, are most frequently applied elements in the finite element method. Their main characteristic is achievement of monotonous convergence of the obtained results increasing the element number by which a certain structure is modeled. This convergence is very often a slow process what requires, in order to obtain an "exact" result, a great number of finite elements modelling the structure.

In the theory of finite elements non-conforming elements are introduced either because of difficulties in achieving the C^1 continuality, that is continuality of both the displacements themselves and their first derivatives at the inter-element boundaries, or for the improvement of finite elements behaviour. So by introducing the incompatibile displacement modes (Wilson and others) the behaviour at bending of the rectangular element is improved for state of the plain stress, as well as eight-nodes of brick element.

In order to decrease the necessary element humber, and at the same time to obtain "exact" results, modification of finite element methode based on applying non-conforming elements which are formed in specific way is considered in this paper. Without diminishing generality of the suggested modified procedure, and because the easier presentation and simple formulas, all considerations in this work will be refere to state of the plain stress case.

Different to the standard formulation, shape functions are introduced through:

$$\bar{N}_i^k(x,y) = (1-\alpha)N_i^k(x,y)+\alpha N_i^r(x,y).$$

Without going, this time, into conciderations which brought up this shape functions formulas, only certain basic ideas will be explained.

Parametar α is dependant on relative element size and the way its definition will be showen here, $N_i^k(x,y)$ is shape function of the basic conforming element attached to i-nodal displacement whilest $N_i^r(x,y)$ is added, incompatible shape function and its formulation will be defined.

The parametar α value goes between 0 and 1 but in practical problems it never takes its extreme values. The value $\alpha = 0$ is achieved when the element is "small", that is when the element grid is very fine, and conforming element behaviour is obtained, what is obvious in the very shape function formulation. In the case of "big" element, that is crude grid of finite elements, the parameter α will be betveen 0 and 1 and according to the previously introduced formulation shape function element is non-conforming. Notious "small-big" are relative in the sence that their are dependant on both the order of interpolating polinome of conforming element and where the element is applied, that is on its capability that at certain size gives "exact" plain displacement state and stress in its range.

Function $N_i^k(x,y)$, are known shape functions of basic conforming element and are not going to be discussed separately. It should be noted that the expression basic element unerstands any of already known elements, with this formulation a new nonconforming element is formed, wich with the basic element has the same shape, same nodale degrees of fredom and partly the same shape functions. This joined, derived, element when decreasing its size, α->0, becaomes its basic element.

Functions $N_i^r(x,y)$, two for each nodal displacement: $N_{iu}^r(x,y)$, $N_{iv}^r(x,y)$, and in the case of the plain stress state corresponds to displacement distrbution on element in case that i-th nodal displacement caused by force which is equal to relevant stiffness coefficient K_{ii}, whilest all other nodal displacements are equal to 0. It is obvious that in that case i-th nodal displacement is equal 1, while the nodal forces achieving this state equal stiffness coefficients K_{ji}. These functions do not achive displacement continuality at inter-element boundaries so the obtained element is nonconforming one.

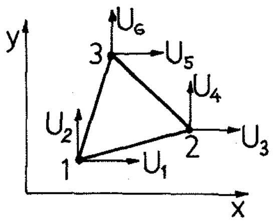
Modification of triangular finite element

Finite element of triangular shape is

well known. Displacement in element are expressed in the form of:

$$\{u\} = \begin{Bmatrix} u(x,y) \\ u(x,y) \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \cdot \begin{bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{bmatrix}$$

where: $N_i = \frac{1}{2A}(a_i + b_i x + c_i y)$ $i=1,2,3$ are known shape function



$$2\Delta = \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}, \quad a_i = x_j y_k - x_k y_j, \quad b_i = y_j - y_k, \quad c_i = x_k - x_j,$$

Fig. 1

and $u(x_i, y_i) = U_{2i-1}$, $v(x_i, y_i) = U_{2i}$, $(i=1,2,3)$ nodal displacements.

Stiffnes matrix and other characteristic of this element are well known and will not be quoted here.

In oreder to define functions $N_i^r(x,y)$, consider now finite element of triangular shape as structure (that is as superelement) modeled throught set of smaller finite elements. Restricted displacement as the applied force in the node, for analized caxes, are shoun on picture 2, where forces F_{3x}, F_{3y},..., may have arbitrary values, subjected thet the problem is in the range of linear elasticity.

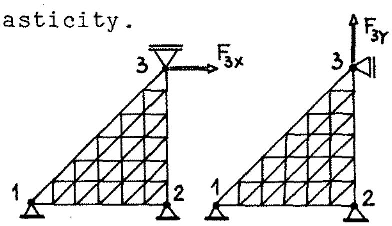


Fig. 2

Using the standard program, displacement & stresses in the superelement and reactions in supports, for shown cases, may be defined. Deviding obtained results by values of the obtained nodal displacement, corresponding to the outher force a normalized displacement distribution is obtained, normalized distribution of stress and normalized values of outher nodal forces system. Normalized values of this nodal displacement is equal to 1 now, whilest the relevant in thus way normalized nodal force is equal to stiffness coefficient K_{ii} (because it represents the force, which causes unit displacement in the node i, and all other nodal displacement equals to 0). Normalized stress distribution now can be defined as stress distribution caused by unit displacement U_i. The stresses are in balance with outher force K_{ii}, as well as with the reactions in the supports, and their values in this case are equal K_{ji}. It should be pointed out that with the finite elements, defined on the base of displacement in general case stresses in element are not in balance with the corresponding nodal forces K_{ji}. Oll these results are obtained by program. So examining rectangular triangle, its division into smaller finite elements is showen in the picture 2, distribution of normalized displacement is obtained, some of them are showen in the figures 3 and 4. This displacement distributions are in fact looked for functions $N_i^r(x,y)$.

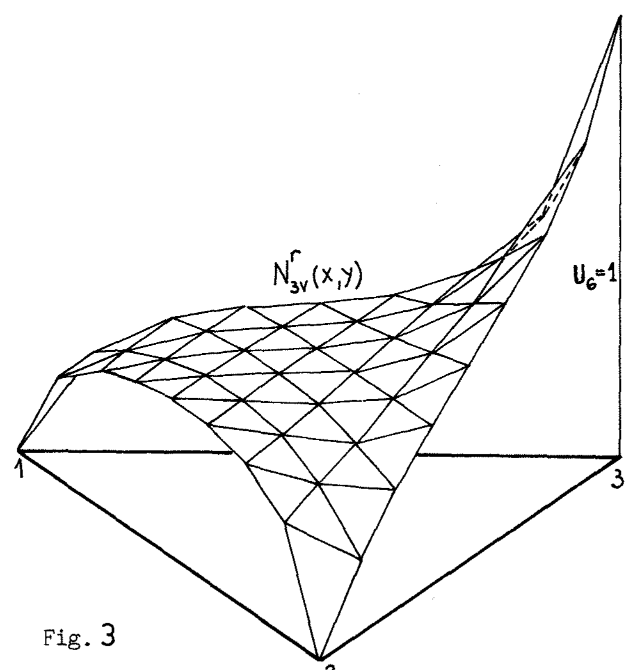


Fig. 3

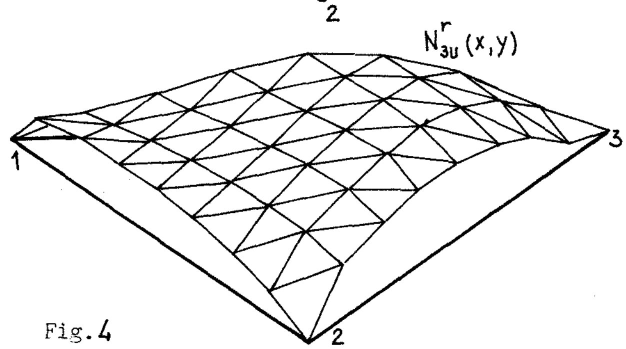


Fig. 4

To form stiffness matrix of modified finite element, we will express displacement on the element throught all nodal displacements by means of modified shape functions. So, based on the previous, we may write:

$$\{u\} = \begin{bmatrix} (1-\alpha)N_1^k+\alpha N_{1u}^r & \alpha\cdot N_{2u}^r & (1-\alpha)N_3^k+\alpha N_{3u}^r & \alpha\cdot N_{4u}^r & (1-\alpha)N_r^k+\alpha N_{5u}^r & \alpha\cdot N_{6u}^r \\ \alpha\cdot N_{1v}^r & (1-\alpha)N_2^k+\alpha N_{2v}^r & \alpha\cdot N_{3v}^r & (1-\alpha)N_4^k+\alpha N_{4v}^r & \alpha\cdot N_{5v}^r & (1-\alpha)N_6^k+\alpha\cdot N_{6v}^r \end{bmatrix} \cdot \begin{bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{bmatrix}$$

that is, after reducing the equotion, the following:

$$\{u\} = (1-\alpha)\begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{bmatrix} + \alpha \begin{bmatrix} N_{1u}^r & N_{2u}^r & N_{3u}^r & N_{4u}^r & N_{5u}^r & N_{6u}^r \\ N_{1v}^r & N_{2v}^r & N_{3v}^r & N_{4v}^r & N_{5v}^r & N_{6v} \end{bmatrix} \begin{bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{bmatrix}$$

where: $N_1^k=N_2^k=N_1$, $N_3^k=N_4^k=N_2$, $N_5^k=N_6^k=N_3$.

Now, element deformation may be expressed in a standard form

$$[B_o] = (1-\alpha)\,[B_o] + \alpha[B_1],$$

where:

$$[B_o] = \begin{bmatrix} \partial/\partial x & 0 \\ 0 & \partial/\partial y \\ \partial/\partial y & \partial/\partial x \end{bmatrix} \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}$$

is a known expression for deformation on a finite element, whilest $[B_1]$ is obtained on the base of the same matrix-differential operator applied only to added (equilibrated) shape functions:

$$[B_1] = \begin{bmatrix} \partial/\partial x & 0 \\ 0 & \partial/\partial y \\ \partial/\partial y & \partial/\partial x \end{bmatrix} \begin{bmatrix} N_{1u}^r & N_{2u}^r & N_{3u}^r & N_{4u}^r & N_{5u}^r & N_{6u}^r \\ N_{1v}^r & N_{2v}^r & N_{3v}^r & N_{4v}^r & N_{5v}^r & N_{6v}^r \end{bmatrix}$$

Stresses may be writen in the form of

$$\{\sigma\}=[D][B]\{U\}=(1-\alpha)[D][B_o]\{U\}+\alpha[D][B_1][U].$$

Element stiffness matrix, as known, is determined by the expression

$$[k^e] = \int_{V^e}[B]^T[D][B]dV^e$$

and now, the following can be written:

$$[k^e]=(1-\alpha)^2\int_{V^e}B_o^TDB_odV^e+\alpha(1-\alpha)\int_{V^e}B_1^TDB_odV^e + \alpha^2\int_{V^e}B_1^TDB_1dV^e.$$

$$k_{oo}^e = \int_{V^e}B_o^TDB_odV^e, \quad k_{11}^e = \int_{V^e}B_1^TDB_1dV^e$$

$$k_{10}^e = \int_{V^e}B_1^TDB_odV^e = k_{01}^e = \int_{V^e}B_o^TDB_1dV^e$$

$$k^e = (1-\alpha)^2k_{oo}^e + 2\alpha(1-\alpha)k_{01}^e + \alpha^2k_{11}^e.$$

The first matrix k_{oo}^e represents the standard stiffness matrix of a basic finite element, which is known or can be determined by standard procedure.

Third matrix k_{11}^e has also been determined numerically at the previous determination of normalized forces applied at the nodes of finite element. So for the given element, taking for example F_{3x}=200 (daN) achieved displacement U_5=0.0070208 (cm) and reactions on the supports are shown in fig 5a. By normalizing these values with obtained displacement stiffness coefficients are determined K_{ji}, shown in the fig 5b. Other coefficients of matrix k_{11}^e are determined in the same way.

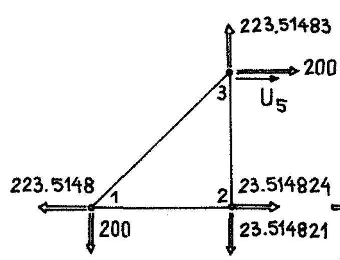


Fig. 5a.

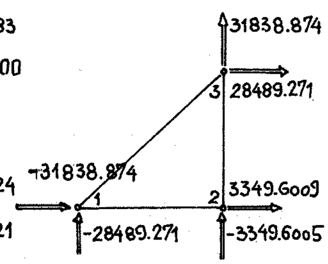


Fig. 5b.

Before determining matrix k_{10}^e and k_{01}^4, notice that these two matrix are transposed one to another and due to it, it is sufficient to examine one of them. If we take to calculate the matrix k_{01}^e which is defined with

$$k_{01}^e = \int_{V^e}B_o^TDB_1dV^e$$

and notice that the matrix B_o is constant (that is all its elements are constants) and may be put in front of the integral sign. Also observe that expression DB_1 represents stresses on an element due to unite displacement, which have been previously determined. As these stresses are constant in each element with which superelement is modeled, the following can be written:

$$k_{01}^{e} = B_o^T \int_{V^e} \{\sigma_1^e\} dV^e$$

and taking care that all elements are equal among themselves, we have:

$$k_{01}^{e} = t \cdot \frac{A_e}{n_e} B_o^T \left(\sum_{e=1}^{n_e} \{\sigma_1^e\} \right),$$

so matrix k_{01}^{e} is obvously obtained in a simple way and the subroutine for its determination is easy to write.

For determining parameter α examine now the structure showen in fig. 6a for which solution is being achived using the standard programe. Let say that the same structure is modeled as in fig. 6b where these elements are formulated as previously determined modified way.

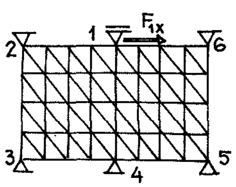


Fig.6a.

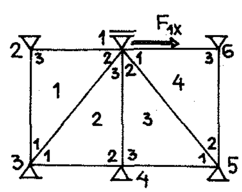


Fig.6b.

As all nodal displacements are equal 0 but the displacement U_1, which is known from previously obtained solution and since the relevant nodal force is also known, follovs that on the base of known procedure for forming the global stiffnes matrix we can directly write:

$$[k_{11}] = (1-\alpha)^2([k_{22}^1]+[k_{33}^2]+[k_{11}^4])_{oo} +$$
$$+2\alpha(1-\alpha)([k_{22}^2 + k_{33}^2 + k_{22}^3]+[k_{11}^4])_{01} +$$
$$+\alpha^2([k_{22}^1]+[k_{33}^2 + k_{22}^3]+[k_{11}^4])_{11} =$$
$$=(1-\alpha)^2[k_{11(oo)}]+2\alpha(1-\alpha)[k_{11(01)}]+\alpha^2[k_{11(11)}] \quad .$$

$$(1-\alpha)^2 \cdot k_{11(oo)}^{11} + 2\alpha(1-\alpha)k_{11(01)}^{11} + \alpha^2 k_{11(11)}^{11} = \frac{F_{1x}}{U_{1x}}$$

where the only unknown α is easy to obtain.

In order to clarify the meaning of parameter α let's examine a simple example shown in Fig. 9. The structure is modeled with

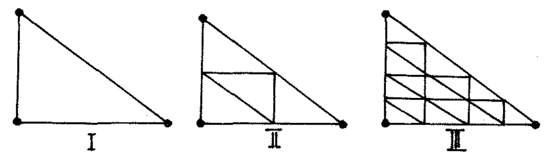


Fig.7

three successive meshs of finite elements of triangular shape, where each of triangular element could be modeled in the following way:

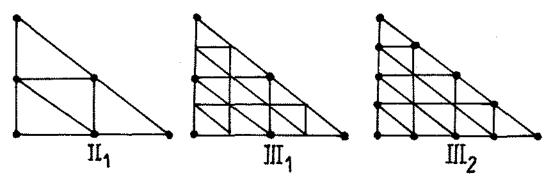


Fig.8

Element I is a known triangular element with a constant state of deformations, whilest the elements II and III are derived superelemerts where the elimination of inner nodal degrees of freedom was performed.

The initial idea was to find out what results would be obtained in case of finite elements being connected through nodes in spots in a global model, and after that according to the W. Dirschmid's idea [Ref.1] make self adaptive programe, which by iterative procedure connects previously reciprocally unconnected nodes in individual edges of these elements and eventually depending on the stress diference value in neighbouring elements going onto the next division.

Explaining the obtained results, their tipical form is shown in Fig. 10, the idea to modify the metode of finite elements was born.

Introducing a new sign II_1 in case where element II is connected in all its nodes, as well as the sign III_1 when the element

III is connected in the nodes in the middle of its edges, that is the sign III_2 when connected in all its nodes along the edges. These elements are shower in Fig. 8.

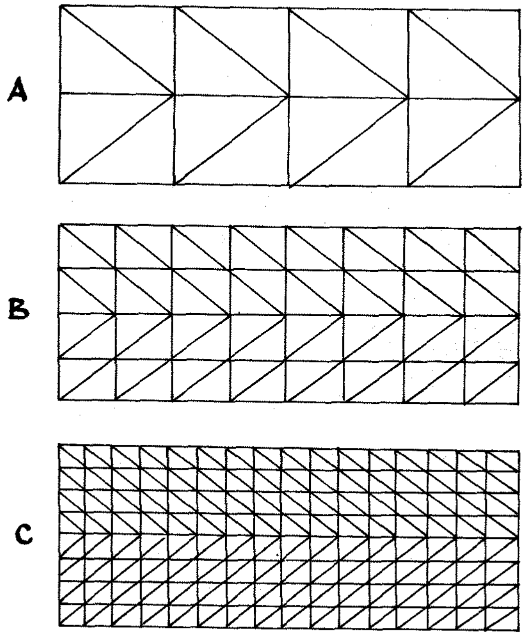


Fig. 9

The combination of these signs and the signs A, B and C for the applied finite elements mesh, whole range of models with different nodal connection degrees on the element boundary is obtained. Tipical set of results is shown in Fig. 10. The results obtained by successivly connected nodes are put together by broken lines, marking the change from one model to another. Dirschmid's iterative procedure "goes" along one of the lines and it is ended when the displacement difference of the disconnected nodes and the relevant nodal displacements as a linear interpolation obtained by displacement at the neighbouring already connected nodes is less then some in advanced defined value.

Instead of successive iterative calculation of ressults which define the broken line this process can be define inadvance. Let the set of finite elements models be so defined that with the decreasing of the element this goes throught a set of N, N-1 .. where N, N-1, are defined successive finer divisions of the element than those showen in the Fig. 8. If it's, at the same time, imagined that with the decrease of element, we go trought the set A, B, C, successively finer meshes of structure, it is obvious from the picture that we are coming closer to the exact solution from the upper side, along the curve A_N - B_{N-1} - C_{N-2} shown on the Fig. 10.

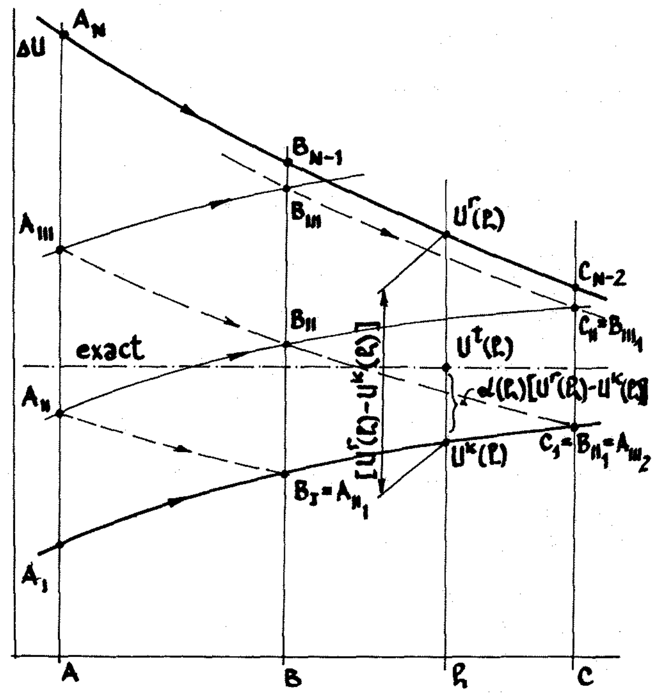


Fig. 10

For each h characterizing the size of the element, there are now two solutions, one on the lower curve, corresponding to the application of conforming elements of the size h and is marked with $U^k(h)$, and other on the upper curve corresponding to the superelement of the same size h. As in this superelement equilibrium conditions at all inner nodes are satisfied and as it is not connected to other elements trought the nodes along the edges, but only at the spots, that's why this element is marked as equilibrized, and the solution on the upper curve is marked with $U^k(h)$. These two solutions limit the exact solution from the both lower and the upper side. Having that in mind it can be writen that for each division h, there is a number $\alpha(h)$ such as that

$$U^t(h) = U^k(h) + \alpha(h)[U^r(h) - U^k(h)] =$$

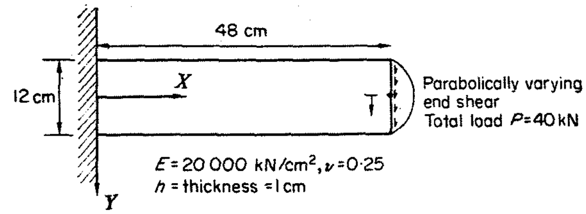
$$= (1-\alpha)U^k(h) + \alpha(h)U^r(h)$$

where $U^t(h)$ is the relevant exact solution. A certain number of global nodes corresponds to each h, it means that we wouid have exact solutions for displacement in same G(h) nodes. The conforming model convergetes to exact solutions in all global nodes too, but the exact solution is obtained only in case of large number of small elements. Nonconforming model, as previously shown, can give exact solutions in limited number of nodes, and in the case of very crude divisions but in that case parameter $\alpha(h)$ must be defined.

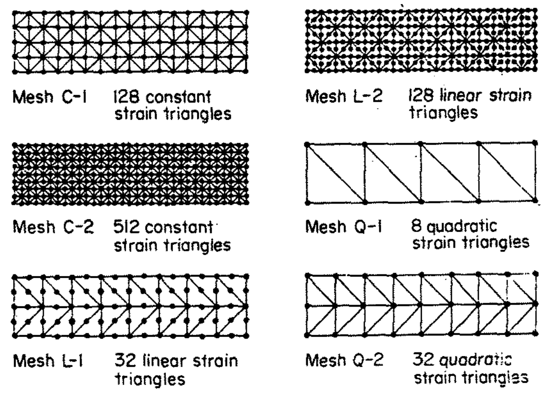
On the base of the previously shown methode for determing parameter α by means of "etalon substructure" it is obvioust that it refers to the groupe of elements, that depends on the configuration of the element mesh and depends on the character of the stress distribution in that part of structure. The standard wing construction, fuselage or tale surfaces in aircraft structures, as the standard loads of these parts, with the standard divisions of these parts on finite elements, what is already proved in practice, enables that once defined parameter α can be used for all simillar constructions. Parameter α is so defined for a certain group of elements and for a certain part of the structure.

In order to show the rate of convergence of the results obtained by applying these elements, we'll take example from Ref. [2].

The same structure is modeled as shown in Fig. 12 stressing that the system has only 16 degrees of treedom, for vertical displacement it's achived at the free end v=0.50205 (cm) as average value of deflections at the nodes 1 and 2 in wich displacements are: V_1=0.4923 (cm) and V_2=0.5118 (cm). The stress in node 8 is determined as average stress value σ_x in elements which are conected through this node and reads: 58276.32 (N/cm²). The value of parameter α = 0.78 is obtained for this case.



Beam and load system



Cantilever beam under end shear load—Triangular meshes

CANTILEVER BEAM: COMPARISON BETWEEN CONSTANT, LINEAR AND QUADRATIC STRAIN MESHES

Deflection and normal stress				
Element	*Mesh*	*Total number of nodal unknowns*	*Tip deflection* v_c (cm)	*Stress* σ_x (N cm⁻²) *at* X = 12 cm, Y = 6 cm
Constant strain triangle	C-1 C-2	160 576	0·458 34 0·512 82	51·225 57·342
Linear strain triangle	L-1 L-2	160 576	0·532 59 0·533 53	59·145 60·024
Quadratic strain triangle	Q-1 Q-2	68 214	0·530 59* 0·532 59	58·973* 59·843
Beam theory (Upper bound for v_c)			0·533 74	60·000

*Average of values at Y = 6 cm and Y = −6 cm

Fig. 11

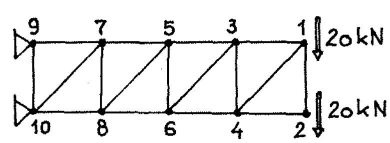


Fig. 12

Concluison

The paper presents a description of the modification of finite element methode.

The sugested modified method has already been used for some practical purposes and there are possibilities of developing it further.

References

[1] W.Dirschmid,

"An Iteration Procedure for Reducing the Expenses of Static, Elastiplastic and Eigenvalue Problems in Finite Element Analyses",

Computer Methods in Applied Mechanics and Engineering 35, pp.15-33, 1982.

[2] C.A. Brebbia, J.J. Connor,

"Fundamentals of Finite Element Techniques",

Butterworths & Co., London, 1973.

[3] A.Samuelsson,

"The Global Constant Strain Condition and the Parch Test"

pp.47-58,

"Energy Methods in Finite Element Analysis"

edited by R.Glowinski, E.Y.Rodin, O.C. Zeinkiewicz, John Wiley & Sons, New York, 1979.

[4] K.P. Jacobsen,

"Fully Integrated Superelements: A Database Approach to Finite Element Analysis",

Computers & structures Vol. 16, No.1-4, pp 307-315, 1983.