ANNALS OF THE UNIVERSITY OF ORADEA
Fascicle of Management and Technological Engineering
ISSUE #3, DECEMBER 2014, http://www.imtuoradea.ro/auo.fmte/

THE FINITE ELEMENT METHOD USED FOR ANALYSIS THE TRIAXIAL STRESS AND STRAIN STATE

Horea HORA[1], Cristina HORA[2]
[1] Faculty of Management and Technological Engineering, University of Oradea, e-mail:horeahora@yahoo.com
[2] Faculty of Energyl Engineering and Industrial Management, University of Oradea, e-mail:chora@uoradea.ro

Abstract **—** The lenses as characteristic for optical systems are volume parts whose analysis of stress and strain state requires a three-dimensional approach. This paper contains a description of the Finite Element Method principle (FEM) in the analysis of triaxial stress and strain state, with specific on tetrahedral linear finite elements, which are suitable to pieces meshing such as the lenses**.**

Keywords **—** Lenses, stress and strain, Finite Element Method

I. INTRODUCTION

FINITE element's method (FEM) has established itself as a modern method of solving a wide range of engineering problems that traditional methods, analytical, are difficult or impossible to solve.

FEM is based on the idea that the continue material structure of real body, having a infinite degree of freedom, can be divided into a finite number of subdomains, with a geometric and physicomechanical own status, and having attached a number, also finished, of freedom degrees.

These subdomains, called finite elements, through reassembling, reconstruct the initial body, and render its reaction with a certain degree of approximation.

Mathematical method of calculation is based on variational principles [1], [2]. In mechanical problems of elasticity [3], it involves a scalar quantity, the Π function, defined by an integral form :

$$\Pi = \int_{\Omega} F\left(u, \frac{\partial}{\partial x} u, \ldots\right) d\Omega + \int_{\Gamma} E\left(u, \frac{\partial}{\partial x} u, \ldots\right) d\Gamma \quad (1)$$

where u represents unknown function,

E, F – operators
Ω, Γ - domains
x, … - independent variables

The solution of continuity problems is a function based on the stationary nature of Π function towards small variations δu. Therefore, a solution to a problem of continuity, has the main phenomenological formula:

$$\delta\Pi = 0 \quad (2)$$

In general, if a problem may be applied to a variational principle, then we can find a function $\underline{u}$, to approximate the u function, in the form:

$$u \cong \underline{u} = \sum N_i a_i \quad (3)$$

where N_i represents shape functions, depending on independent variables, and on all or only some of the unknown measures a_i..

Applying (3) leads to a system of equations with a_i unknown.

Stationarity finding methods using functions that depend on a_i parameters are named after Rayleigh and Ritz.

Solving an application through FEM follows a route that includes the following steps [1], [2] :
1) meshing analysis domain
2) elements' equation formatting and their assembly to the structure of the equation system
3) solving the equation system
4) *postprocessing results*

II. MESHING ANALYSIS DOMAIN

By meshing, real solid is divided into simple subdomains which, by reassembling, render a high approximation, the form from which it started. These elements become, through mathematical modelling, finite elements.

Finite elements may be linear, two or three-dimensional, depending on the requirements of the application.

The most simple spatial forms, suitable for meshing bodies with curved surfaces, such as lenses, are prismatic or tetrahedral elements [4], [5], [6]. From a mathematical perspective, in order to preserve the linear element, splitting prism in tetrahedral elements is applied. Triangle based prism generates three tetrahedra, (Fig. 1.) and the trapezoid prism decomposes into five tetrahedra (Fig. 2.).

Prism characteristic parameters result easy, by averaging the quantities corresponding to tetrahedra components. So, for brick items, computer programs deliver sizes of nodes and a central value.

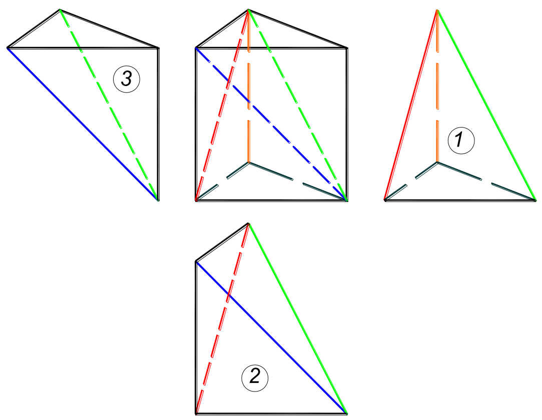

Fig. 1. Decomposition of brick elements having triangle tetrahedra base

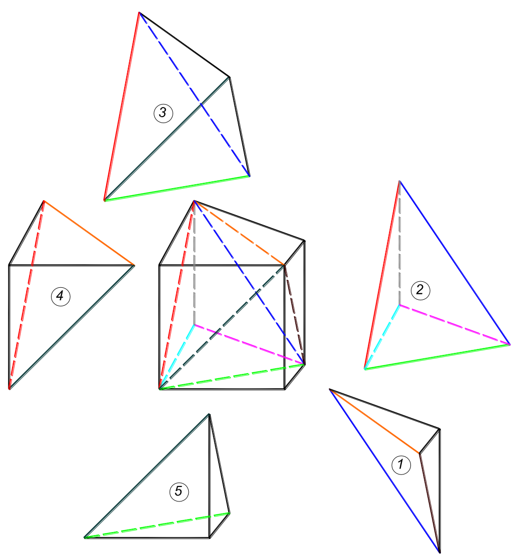

Fig.2 Decomposition of brick elements having quadrilateral tetrahedra base

III. Establishment Of Finite Element Equations And Their Assembly In Equation System Structure

Tetrahedral finite elements with four nodes is a linear element defined by its local coordinates reported to the i, j, p, m nodes or by global coordinates reported to an xyz triortogonal system, exterior to the structure (Fig. 3.).

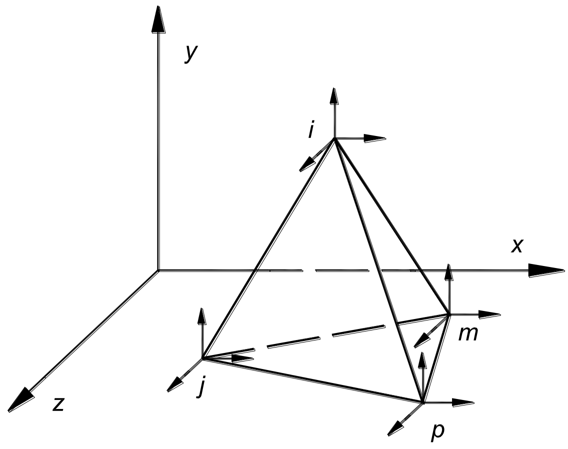

Fig. 3. Local and global coordinate system of tetrahedral element

In applications, where stress and strain are analyzed, the main unknown functions are the displacements [3], [6], [7], [8]. A vector of displacements is defined on a {f} element, and a vector of nodal displacements $\{d_e\}$ (4 and 5):

$$\{f\}=\begin{Bmatrix} u \\ v \\ w \end{Bmatrix} \tag{4}$$

$$\{d_e\}=\begin{Bmatrix} u_i \\ v_i \\ w_i \\ u_j \\ v_j \\ w_j \\ u_m \\ v_m \\ w_m \\ u_p \\ v_p \\ w_p \end{Bmatrix} \tag{5}$$

Displacement functions u, v, w are approximated on the element by linear interpolation functions:

$$\begin{cases} u = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z \\ v = \alpha_5 + \alpha_6 x + \alpha_7 y + \alpha_8 z \\ w = \alpha_9 + \alpha_{10} x + \alpha_{11} y + \alpha_{12} z \end{cases} \tag{6}$$

In (6) u, v, w displacements are replaced by nodal values, and x, y, z coordinates with the corresponding nodes values. A 12 equation system results from here, having the unknown coefficients $\alpha_1 \ldots \alpha_{12}$.

These are called generalized coordinates and have no physical significance. In order to determine elementary displacements, nodal displacements must be determined.

The displacement shape functions on the element, reported to nodal displacements, are represented in 7, 8 and 9:

$$u = \frac{1}{6V}\begin{Bmatrix} (a_i + b_i x + c_i y + d_i z)u_i + (a_j + b_j x + c_j y + d_j z)u_j + (a_p + b_p x + c_p y + d_p z)u_p + \\ + (a_m + b_m x + c_m y + d_m z)u_m \end{Bmatrix} \tag{7}$$

$$u = \frac{1}{6V}\begin{Bmatrix} (a_i + b_i x + c_i y + d_i z)v_i + (a_j + b_j x + c_j y + d_j z)v_j + (a_p + b_p x + c_p y + d_p z)v_p + \\ + (a_m + b_m x + c_m y + d_m z)v_m \end{Bmatrix} \tag{8}$$

$$u = \frac{1}{6V}\begin{Bmatrix} (a_i + b_i x + c_i y + d_i z)w_i + (a_j + b_j x + c_j y + d_j z)w_j + (a_p + b_p x + c_p y + d_p z)w_p + \\ + (a_m + b_m x + c_m y + d_m z)w_m \end{Bmatrix} \tag{9}$$

where V is the volume of the tetrahedron.
$a_i \ldots a_m$, $b_i \ldots b_m$, $c_i \ldots c_m$, $d_i \ldots d_m$ - coefficients calculated depending on the nodes' coordinates.

Element displacement functions can be written as in (10):

$$\begin{cases} u = N_i u_i + N_j u_j + N_p u_p + N_m u_m \\ v = N_i v_i + N_j v_j + N_p v_p + N_m v_m \\ w = N_i w_i + N_j w_j + N_p w_p + N_m w_m \end{cases} \tag{10}$$

where:

$$\begin{cases} N_i = a_i + b_i x + c_i y + d_i z \\ N_j = a_j + b_j x + c_j y + d_j z \\ N_p = a_p + b_p x + c_p y + d_p z \\ N_m = a_m + b_m x + c_m y + d_m z \end{cases} \tag{11}$$

N_i, N_j, N_p, N_m functions are called interpolation functions and have value 1, in the nodes with the same index. In the other nodes they are null (Example: in i node, N_i=1. And in j, p and m nodes N_i=0).)

In order to determine nodal displacements, with elasticity problems, a variational formulation applied to energy methods is used. For this, Π functional is defined:

$$\Pi = U - W \tag{12}$$

where: U represents strain energy, W - the mechanical work of external forces.

For one element we can write:

$$U_e = \int_{V_e} \left(\frac{1}{2} \{\varepsilon\}^T [E] \{\varepsilon\} + \{\varepsilon\}^T \{\sigma_0\} \right) dV \tag{13}$$

$$W_e = \int_{V_e} \{f\}^T \{F\} dV + \int_{S_e} \{f\}^T \{Q\} dS + \{d_e\}^T \{p_n\} \tag{14}$$

where $\{\varepsilon\}$ is specific displacement vector;
[E] - elasticity matrix;
$\{\sigma_0\}$ - initial tension vector;
{f} - elementary displacement vector;
{F} - volumetric forces vector, acting on the element;
{Q} - surface force vector, acting on the element;
$\{p_n\}$ - concentrated force vector, acting on the element;
$\{d_e\}$ - nodal displacement of the element vector.

Specific strains and elementary displacements are expressed depending on nodal displacements:

$$\{\varepsilon\} = [B]\{d_e\} \tag{15}$$

$$\{f\} = [N]\{d_e\} \tag{16}$$

For a system that contains M elements, potential energy will be:

$$\Pi = \sum_{1}^{M} \Pi_e \tag{17}$$

Minimizing the Π functional equals, mathematically, the requiring of cancellation condition of partial derivatives related to d_i displacement of the element:

$$\frac{\partial \Pi}{\partial d_i} = 0, \quad i = \overline{1,n} \tag{18}$$

where n represents degrees of freedom number of the system and is the product of the number of nodes and the number of degrees of freedom per node.

A system of M equations results:

$$\left(\sum_1^M \int_{V_e} [B]^T [E][B] dV\right)\{d\} = \sum_1^M \left(-\int_{V_e} [B]^T \{\sigma_0\} dV + \int_{V_e} [N]^T \{F\} dV + \right.$$
$$\left. + \int_{S_e} [N]^T [Q] dS \right) + \{P\} \qquad (19)$$

where $\{d\}$ is nodal displacement vector of the following structure:

$$\{d\} = \sum_1^M \{d_e\} \qquad (20)$$

$\{P\}$ - nodal forces vector of the structure:

$$\{P\} = \sum_1^M \{p_n\} \qquad (21)$$

Stiffness matrix of the [k] element and of system [K] is defined:

$$[k] = \int_{V_e} [B]^T [E][B] dV \qquad (22)$$

$$[K] = \sum_1^M \int_{V_e} [B]^T [E][B] dV \qquad (23)$$

Finally we obtain the canonical form of the finite element matrix equation (the fundamental equation):

$$[K]\{d\} = \{R\} \qquad (24)$$

where $\{R\}$ represents, in the most concise form, the volume requests vector, of surface and concentration.

For the finite tetrahedral element, previous general considerations are particularized as follows.

Starting from elementary displacement vector, (25), which is expressed depending on nodal displacements, using N formed functions, previously determined:

$$\{f\} = \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = [N]\{d_e\} = [N]\begin{Bmatrix} u_i \\ v_i \\ w_i \\ u_j \\ v_j \\ w_j \\ u_p \\ v_p \\ w_p \\ u_m \\ v_m \\ w_m \end{Bmatrix} \qquad (25)$$

where N shape function matrix is:

$$[N] = \begin{bmatrix} N_i & 0 & 0 & N_j & 0 & 0 & N_p & 0 & 0 & N_m & 0 & 0 \\ 0 & N_i & 0 & 0 & N_j & 0 & 0 & N_p & 0 & 0 & N_m & 0 \\ 0 & 0 & N_i & 0 & 0 & N_j & 0 & 0 & N_p & 0 & 0 & N_m \end{bmatrix} \qquad (26)$$

The matrix of the unit deformations can be written as:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \end{Bmatrix} = [B]\{d_e\} \qquad (27)$$

[B] is called matrix transformation of displacements in unit deformations and has the following form:

$$[B] = \begin{bmatrix} b_i & 0 & 0 & b_j & 0 & 0 & \ldots \\ 0 & c_i & 0 & 0 & c_j & 0 & \ldots \\ 0 & 0 & d_i & 0 & 0 & d_j & \ldots \\ c_i + b_i & 0 & 0 & c_j + b_j & 0 & 0 & \ldots \\ 0 & d_i + c_i & 0 & 0 & d_j + c_j & 0 & \ldots \\ 0 & 0 & d_i + b_i & 0 & 0 & d_j + b_j & \ldots \end{bmatrix} \qquad (28)$$

Elasticity matrix [E] has the following form:

$$[E]=\frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix}1-\nu & \nu & \nu & 0 & 0 & 0\\ 0 & 1-\nu & \nu & 0 & 0 & 0\\ 0 & 0 & 1-\nu & 0 & 0 & 0\\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0\\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0\\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2}\end{bmatrix} \quad (29)$$

Stress vector is written using Hooke's law, generalized:

$$\{\sigma\}=\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{zx}\end{Bmatrix}=[E]\{\varepsilon\} \quad (30)$$

Stiffness matrix of finite element becomes:

$$[k]=\int_{V_e}[B]^T[E][B]dV \quad (31)$$

Acting forces vector on the element can be:

$\{f_0\}_e$ - forces vector due to initial stress:

$$\{f_0\}_e=-\int_{V_e}[B]^T\{\sigma_0\}dV \quad (32)$$

$\{f_m\}_e$ - volumetric forces vector (mass):

$$\{f_m\}_e=\int_{V_e}[N]^T\{F\}dV=\int_{V_e}[N]^T\begin{Bmatrix}F_x\\ F_y\\ F_z\end{Bmatrix} \quad (33)$$

In the above expressions, such integrals are written:

$$I=\int_{V_e}L_1^{\alpha}L_2^{\beta}L_3^{\gamma}L_4^{\delta}dV \quad (34)$$

Their value equals with:

$$I=\frac{\alpha!\beta!\gamma!\delta!}{(\alpha+\beta+\gamma+\delta)}6V \quad (35)$$

$\{f_S\}_e$ - Surface forces vector:

$$\{f_S\}_e=\int_{S_e}[N]^T[Q]dS=\int_{S_e}[N]^T\begin{Bmatrix}Q_x\\ Q_y\\ Q_z\end{Bmatrix}dS \quad (36)$$

$\{r\}_e$ - concentrated forces vector:

$$\{r\}_e=\begin{Bmatrix}r_x\\ r_y\\ r_z\end{Bmatrix} \quad (37)$$

A system of finite elements equations is written, in canonical form:

$$[k]\{d\}_e=\{f\}_e \quad (38)$$

The matrix and the elements' vectors in (38) have the following dimensions:

[k] - 4x3 (nodes number x degrees of freedom per node number)

$\{d\}_e$ - 1x12 (12 = shifting components' number x nodes number)

$\{f\}_e$ - 1x12 (12 = force components' number x elements' nodes number)

Having determined the finite elements' equations of the structure, we proceed to their assembly, as opposed to meshing.

Geometrically, the solid structure from which we started is recovered, and mathematically, a global model of the system is resulting.

Assembly is made separately, on stiffness matrix, displacements and forces, following two operations:

1) expansion of elementary matrices

2) adding expanded matrices.

Stiffness matrix of the system is a square matrix with nxn dimensions (n – networks' nodes number x degrees of freedom per node number).

Expanded matrices of nxn dimensions, are obtained by replacing k_{ij} (3x3) matrices in the system's matrix, based on a connection matrix. Other expanded matrixes' elements are considered to be null.

The systems' stiffness matrix results from the sum of expanded matrixes.

The systems' displacement vector U (1x3M),, includes all components of nodal displacements:

$$\{U\}=\begin{Bmatrix}u_1\\ v_1\\ w_1\\ \vdots\\ u_M\\ v_M\\ w_M\end{Bmatrix} \quad (39)$$

Nodal forces vector is obtained the same way :

$$\{\mathbf{R}\} = \begin{Bmatrix} \mathbf{R_{x1}} \\ \mathbf{R_{y1}} \\ \mathbf{R_{z1}} \\ \vdots \\ \mathbf{R_{xM}} \\ \mathbf{R_{yM}} \\ \mathbf{R_{zM}} \end{Bmatrix} \tag{40}$$

The following matrix equation results:

$$[\mathbf{K}]\{\mathbf{U}\}=\{\mathbf{R}\} \tag{41}$$

In this equation, the boundary conditions are introduced. These involve, in general, the null values of displacements or rotations.

In the stiffness matrix of the system, the rows and columns of the known movements are canceled. For example, if the displacement value of u_i is known, (41) will be written as:

$$\begin{bmatrix} k_{1,1} & \cdots & k_{1,i-1} & k_{1,i+1} & \cdots & k_{1,n} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ k_{i-1,1} & \cdots & k_{i-1,i-1} & k_{i-1,i+1} & \cdots & k_{i-1,n} \\ k_{i+1,1} & \cdots & k_{i+1,i-1} & k_{i+1,i+1} & \cdots & k_{i+1,n} \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ k_{n,1} & \cdots & k_{n,i-1} & k_{n,i+1} & \cdots & k_{n,n} \end{bmatrix} \begin{Bmatrix} u_1 \\ \vdots \\ u_{i-1} \\ u_{i+1} \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} R_1 \\ \vdots \\ R_{i-1} \\ R_{i+1} \\ \vdots \\ R_n \end{Bmatrix} - u_i \begin{Bmatrix} k_{1,i} \\ \vdots \\ k_{i-1,i} \\ k_{i+1,i} \\ \vdots \\ k_{n,i} \end{Bmatrix} \tag{42}$$

In this way the system's stiffness matrix size and vector length is reduced. Formally, the structure of the reduced form equation is written as:

$$[\mathbf{K_r}]\{\mathbf{U_r}\}=\{\mathbf{R_r}\} \tag{43}$$

IV. Solving The System Equations And Post Processing The Results

Solving the linear system of equations can be done by various direct, iterative or accurate methods. Using the Cholesky factorization method, for example, the coefficient matrix of the system is converted to the product of two triangular matrixes, which enable the easy, direct sequence of an unknown, one by one.

Values are obtained by solving the system nodal displacements [9], [10].

By post-processing we can determine all stress and strain state of the analyzed structure

V. Conclusion

Solving finite element elasticity requires specialized software. Automatic calculation is essential, given that the number of elements varies between several hundred and several thousand, generating systems containing tens or hundreds of thousands of equations.

The presentation of post-processing results, according to the used computer program, allows storing data as numerical values or listing multiple graphic shapes, having attached chromatic scale, which is much more intuitive.

Choosing the tetrahedral finite element as being the most suitable in terms of geometric shape and mathematical modeling for meshing three-dimensional components like lenses .

References

[1] Okruhlik, M., *Finite Element Homework and Programming Applications*, IFToM, p. 3161-3165, Politehnica di Milano, 1995
[2] Zienkiewicz, O.C., *The Finite Element Method*, McGraw-Hill Co., 1977
[3] Dumitru, I., Neguţ, N., "Elasticity, plasticity and material strenght, vol.I, Politehnica Publishing, 2003 "
[4] Jones, R.V *Instruments and Experiences*, John Wiley, New York, 1988
[5] Kingslake, R., "*Lens design fundamentals*", Academic Press, N.Y.,1978
[6] Shannon, R.R., *Optical Specification*, in Handbook of Optics, vol.I, ch.34, McGraw Hill Inc., NY,1995
[7] O'Shea, D.C., Harrigan, E., *Aberration Curves in Lens Design*, in Handbook of Optics, vol.I, ch. 33, McGraw Hill Inc., NY,1995
[8] Lytle, J.D., *Polymeric Optics*, in Handbook of Optics, vol.II, ch. 33, McGraw Hill Inc., NY,1995
[9] Gruescu, C., Strauţi Negru, G., Nicoara I., "*Aspects concerning the influence of execution errors of optical components upon the image quality*," The VIth international Conference on Precison Mechanics and Mechatronics COMEFIM-6, Brasov, 2002, volume 1-20a, p 281-284
[10] Gruescu C., Balaban, G., "*Analyse of transverse spherical aberration of a tilted lens*", The 8.th Symposium on Mechanisms and Mechanical Transmissions with international participation, Timişoara, 2000, p.143-148