Computer Assisted Mechanics and Engineering Sciences, **13**: 235–245, 2006.

A technique of stretched meshes in the modelling of complex 3D structures in Finite Elements Analysis (FEA) applications

A. Smykla
Rzeszów University of Technology
Department of Computer Science
ul. W. Pola, 235-959 Rzeszów
e-mail: asmykla@prz.rzeszow.pl

(Received April 4, 2005)

The paper presents the computational approach to the process of creation of complex three-dimensional finite elements models. The data are automatically received by the measuring unit and sent directly to the computer. The process of numerical treatment of measuring points' coordinates is shown. As an example of above mentioned method, the construction of human femoral head model is demonstrated. Finally the geometrical model of the object, which can be used by PATRAN system is obtain. Paper presents results of numerical calculations and compares it with a photoelastic experiment.

1. INTRODUCTION

In the recent years, numerous CAD applications have been developed intended for use in the creation of analytical models used the finite elements analysis (FEA). Current software provide the user with tools having a graphical interface alleviating the need for painstaking definition of model details, like numbering of nodes or their connections. Typically, the creation process of an analytical model can be described by following steps:

1) creating the geometry using so called primitives (points, curves, surfaces, solids),

2) creating a mesh of finite elements (meshing) over the created geometry,

3) defining the material constants (assigned either to geometry or to elements),

4) defining the boundary conditions (restrains, forces, contacts),

The meshing on previously created geometry has been presented by numerous papers: [3, 7, 9, 10, 13–15]. For typical surfaces and solids, the meshing is well described and, therefore, it will not be considered in this paper.

Instead, special consideration will be given to the creation of numerical models of structures having very complex geometry. Some natural objects can be a good example of such structures. In recent years, the raise of interest in such structures has been noticed [1, 2, 4, 8, 12].

This paper is illustrated with an analytical example of this kind, too.

In the presented example, the information about geometry which was input into the computer was obtained from the Vita Mot co-ordinate measuring machine, make of the Vista company.

The process of defining the co-ordinates of border points on the measuring station is depicted in Fig. 1.

The co-ordinates of border points ($\mathrm{Brz}_1 \ldots \mathrm{Brz}_n$) received from the measurements are shown in Fig. 2.

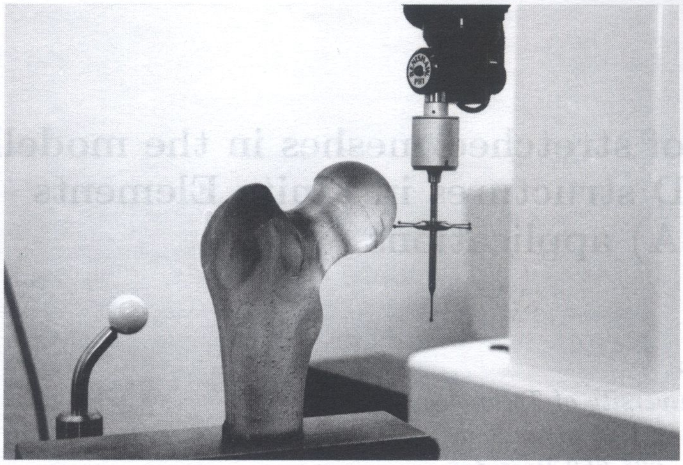

Fig. 1. Details of geometry scanning of a femoral head casting

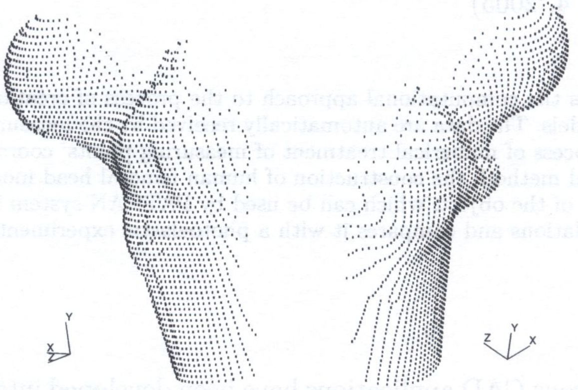

Fig. 2. Mesh of points obtained from co-ordinate measuring machine measurements (for clarity, the left and right halves of the model are shown separated)

The acquired in this way data makes it possible to develop an approximate geometric model using some of the measurement points as input data. An example of such model is shown in Fig. 3.

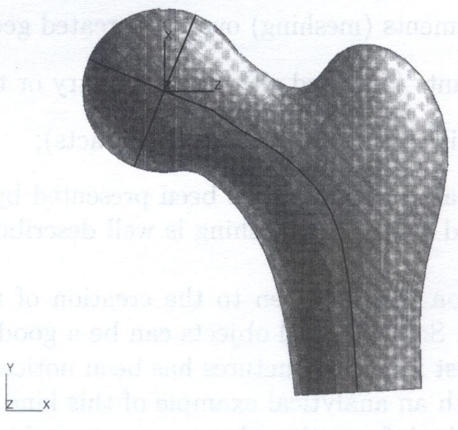

Fig. 3. Geometry of the femoral with adjacent part; approximate model [4]

The model was developed using the pre-processor of PATRAN program in such a way as to make the automatic meshing possible.

This method of model creation is only possible by assuming considerable simplifications and it demands high skill and much imagination from the user. And yet, the modelling is not always possible and limited rather to convex solids of low complexity of shapes of the side surface.

Alas, the model created in such a way lacks a number of essential details.

It is, however, worth mentioning that by using modern software it is possible nowadays to input the geometry information in the form of mathematical equations of the side surface. Unfortunately, the transition from the geometric model to the FEA model is not always that simple or even possible at all.

Paper [7] describes the development process of an FEA model using the voxel representation. Let's have a closer look at this issue. The voxel representation of solids is used to render very complex geometric structures in the computer graphics. The idea of this representation is shown in Fig. 4.

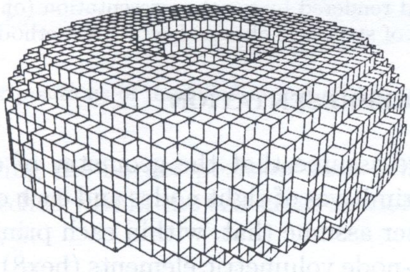

Fig. 4. Torus rendered by voxel representation [6]

It consists in the filling the space occupied by an object with elements of equal size, e.g. cubes as shown in Fig. 4. With a high concentration of the filling elements, the observer can see quite a smooth figure.

This representation is often used for presentation of data received from sources like CAT or during animation creation. The filling composed of 4-node elements which, at the same time, constituted the finite element mesh, was employed by the authors of paper [7]. Figure 5 shows an example taken from the mentioned paper.

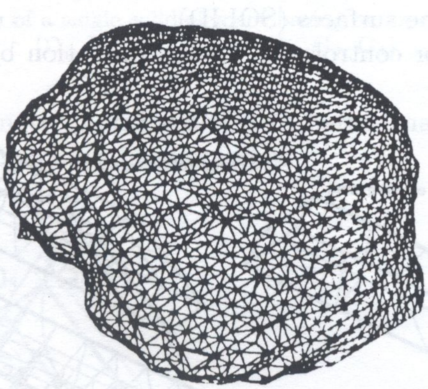

Fig. 5. Voxel representation of brain tumor

Figure 6 shows a model of the femoral head created by this technique; for filling, cube elements were used.

In practice, however, the use of such a structure in tensile calculations is rather limited. Typically, such model has many offsets and step-like surface irregularities resulting in an unsatisfactory outcome of the stress analysis.

The present paper describes an alternative solution for creating geometrically complex models by using a stretched mesh method.

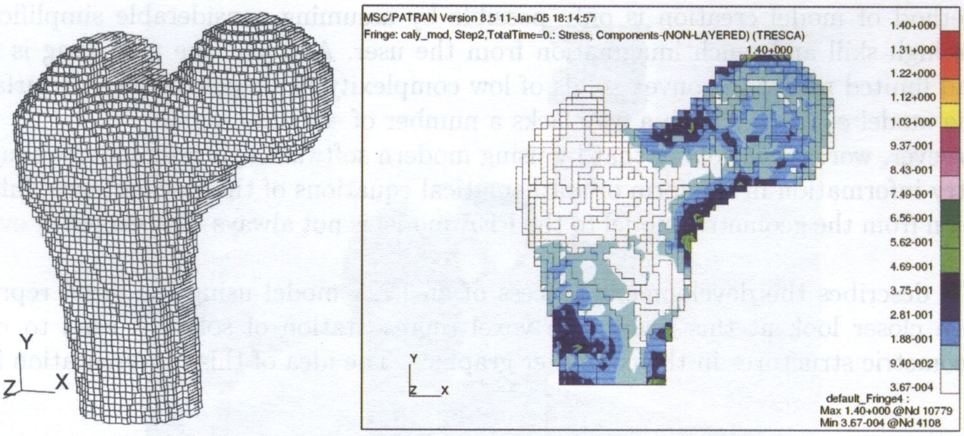

Fig. 6. Model of the femoral head rendered by voxel representation (on the left) and computational model of structure obtained by FEA method

2. DESCRIPTION OF MODEL REPRESENTATION

For the model representation, we assume that the geometry of a complex solid will be split into a number of simpler volumetric primitives of eight nodes and each of the primitives shall be subjected to the meshing process. We further assume that, within each primitive, the meshing process should enable creation of a mesh of eight-node volumetric elements (hex8). Consequently, a proper structure of the geometric primitives would be necessary. If we drop this assumption, the possibility of special meshing using four-node elements (tet4) still will be left. However, since any space can be filled using these elements, this case is less interesting.

The CAD/CAM software nowadays is often provided with script languages which enable to control the operation of the application and to automatise the process of analytical model creation.

For example, using the NASTRAN system pre-processor, the creation of such primitive can be accomplished in several steps:

1. Defining the points (POINT).
2. Defining the curves on specific points (CURVE).
3. Stretching the surface over specific curves (SURFACE).
4. Stretching the solid over the surfaces (SOLID).

The NASTRAN pre-processor controls the system operation by means of an internal language (PCL) [10].

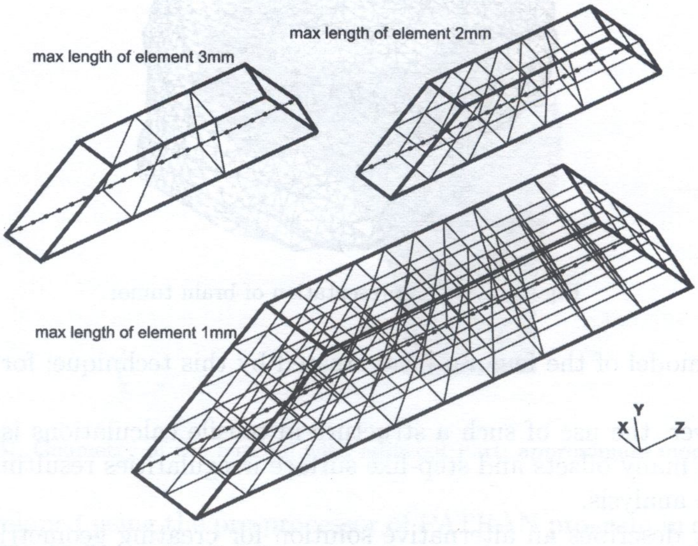

Fig. 7. Examples of divisions into elements (automatic meshing)

By properly assuming the maximal size of the finite elements, the resultant solid can be next subjected to the process of automatic meshing. Figure 7 shows examples of divisions into elements.

In our case, the creation process of volumetric primitives will be carried out by intersecting the modelled structure by a number of parallel planes. For each plane, the border position should be defined, as illustrated in Fig. 8.

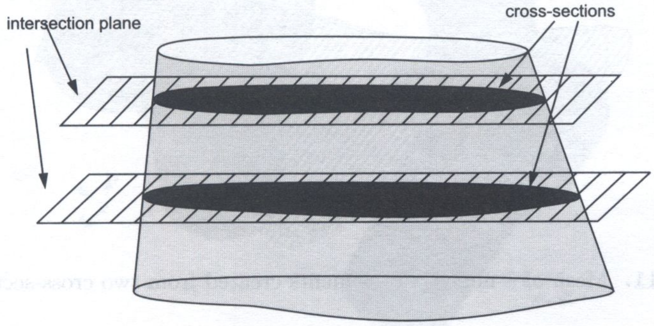

Fig. 8. Defining the border positions of modelled object within two successive cross-sections

The very mechanism of acquiring the border position data will be discussed later in the paper. Next, a mesh of solids will be stretched within the two cross-sections. The idea of creation of a single solid element is illustrated in Fig. 9.

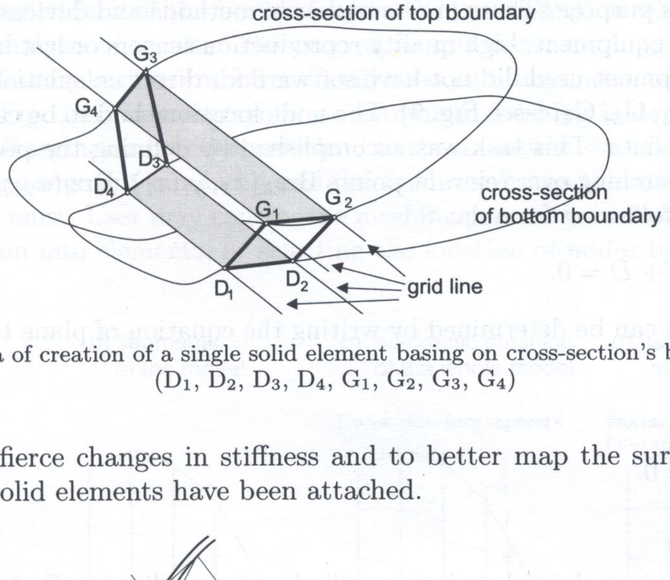

Fig. 9. Idea of creation of a single solid element basing on cross-section's border points (D_1, D_2, D_3, D_4, G_1, G_2, G_3, G_4)

In order to avoid the fierce changes in stiffness and to better map the surface for the outermost solids, the additional solid elements have been attached.

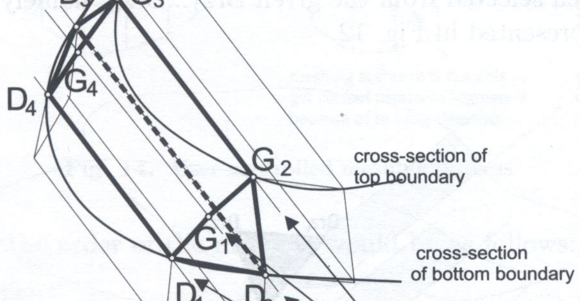

Fig. 10. The location of the grid lines for elements filling the borders of the model

The position of the parallel straight lines $\overline{G_2G_3}$, $\overline{G_4G_1}$, $\overline{D_2D_3}$, $\overline{D_3D_2}$ should be such as to obtain satisfactory approximation of the border. For relatively "dense" division, the border is rendered quite truly. Figure 11 shows an example of such modelling.

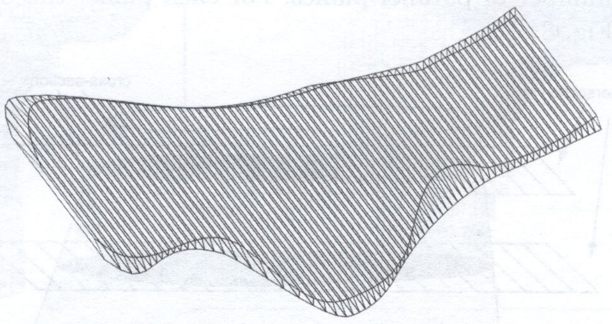

Fig. 11. Mesh of 8-node solid elements created from two cross-sections

3. EXAMPLE OF MODEL REPRESENTATION

As an example of representation of a natural structure, the creation of a numerical model of the human femoral head will be presented. Defining the positions of model's borders becomes an indispensable procedure in the modelling process. Various scanning devices which can feed the acquired information about location of the borders directly into the computer system are used more and more frequently for this purpose. There are a number of methods and devices which can accomplish these tasks, being laser equipment, high quality reproduction sensors or last but not least, the CAT.

The measuring equipment used did not have software for direct calculation of locations of nodes (D_1, D_2, D_3, D_4, G_1, G_2, G_3, G_4, – see Fig. 9). The node locations had to be calculated only from the obtained measurement data. This task was accomplished by defining the positions of the nodes of interest by means of stretching over relevant points $\mathrm{Brz}_k(x_1, y_1, z_1)$, $\mathrm{Brz}_l(x_2, y_2, z_2)$, $\mathrm{Brz}_m(x_3, y_3, z_3)$ a surface given by the following formula:

$$A \cdot x + B \cdot y + C \cdot z + D = 0. \tag{1}$$

The A, B, C, D factors can be determined by writing the equation of plane through 3 points: Brz_k, Brz_l, Brz_m:

$$\begin{vmatrix} x & y & z & 1 \\ x_1 & y_1 & z_1 & 1 \\ x_2 & y_2 & z_2 & 1 \\ x_3 & y_3 & z_3 & 1 \end{vmatrix} = 0. \tag{2}$$

Then, the unknown z is evaluated against specific (x, y) co-ordinates. Earlier, proper points Brz_k, Brz_l, Brz_m should have been selected from the given $\mathrm{Brz}_1 ... \mathrm{Brz}_n$, namely those nearest the node of interest. This situation is presented in Fig. 12.

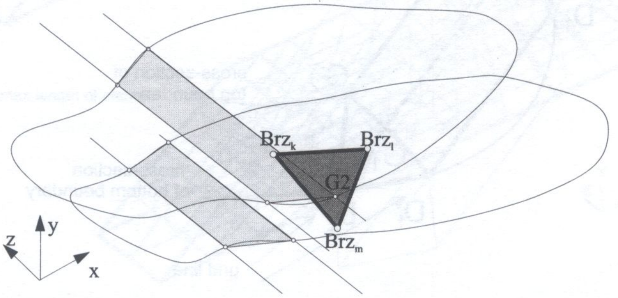

Fig. 12. Defining the position of node G_2 basing on measurement points Brz_k, Brz_l, Brz_m

Figure 13 shows the grid of calculated nodes and the mesh of solid elements spread over the nodes.

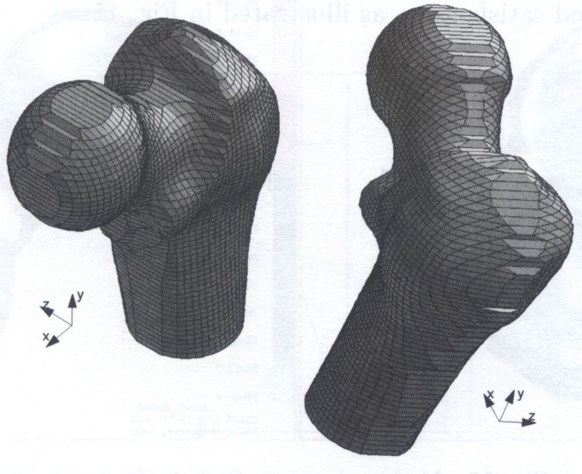

Fig. 13. Element mesh obtained by connecting the node grid.

In the process of division of geometric solids into a mesh of finite elements, the consistency in mesh patterns on borders is essential. In some cases, one can rely on the automatic pre-processor which does the job for the engineer. In other cases it may be helpful to add more divisions of the geometric elements before the automatic meshing process. In the example above, the latter operation was applied to surface of the bone's longitudinal cross-section which, later on, was used to make comparative analysis of results. The process of automatic meshing of the whole structure not always brings satisfactory results and often it will need an intervention of the user. For example, the division of figure (Fig. 14a) into segments for automatic meshing is depicted in Fig. 14b. The automatic meshing of all segments together may lead to the undesired situation presented in Fig. 14c (the mesher started from segment number 4 and the final result was the misfit mesh). In such case a user intervention is a must. User may control the meshing process by selecting the proper order of the segments for division into elements, or selecting the location of nodes by determining so-called mesh seed.

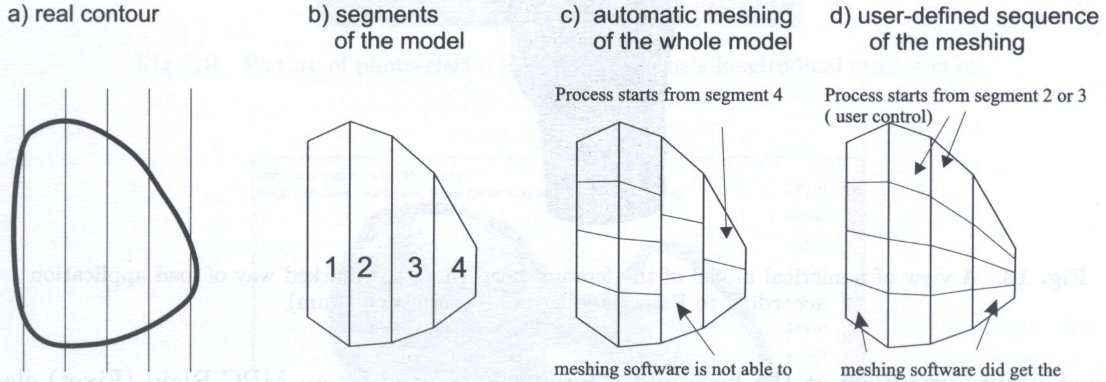

Fig. 14. User-controlled meshing process

For example presented, the order of the meshing would be as follows:

1) segments 2 and 3 together,

2) segment 1,

3) segment 4.

The order should be determined by trial and error method however, its effectiveness will depend on the capabilities of the specific meshing software. In the described example, however, a few steps of automatic meshing proved satisfactory, as illustrated in Fig. 15.

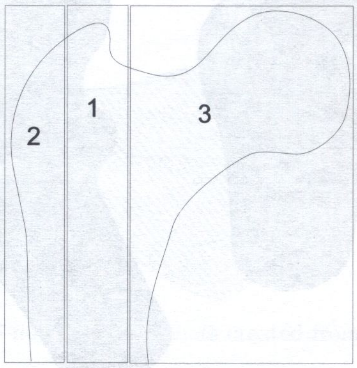

Fig. 15. Meshing sequence for bone fragments

The meshing carried out in the widest place (1) where the solid elements closely resemble cubicoids results in quite regular mesh of finite elements. Taking into account the resultant meshes in step (2) and (3), the pre-processor carried out meshing for the whole structure correctly.

The numerical model was created using the MSC PATRAN system; the resultant analytical diagram and the relevant stress distribution corresponding to the upright human body position according to Pauwels' diagram are presented in Fig. 16.

Fig. 16. A view of numerical model of the femoral head (note the marked way of load application according to Pauwels' diagram – grid every 1 mm)

The structure was fixed at the base and a force was exerted by an MPC Rigid (Fixet) element having six degrees of freedom. Secondary nodes were defined within the head.

The complete model consisted of 86442 finite elements Hex8; additional plane was defined inside the model in order to analyse the stress distribution. The calculations were done using MSC/AFEA program. The TRESCA stress distribution are shown in Fig. 17.

For model verification, a photo-elastic model with frozen stresses was produced. Figure 18 shows the isochromatic lines observed on the analysed axial cross-section.

The photo-elastic model was made of epoxy resin. The calculations, too, were made for the material having constants $E = 2300$ [MPa] and $v = 0.42$ in order to be able to make both qualitative and quantitive comparisons.

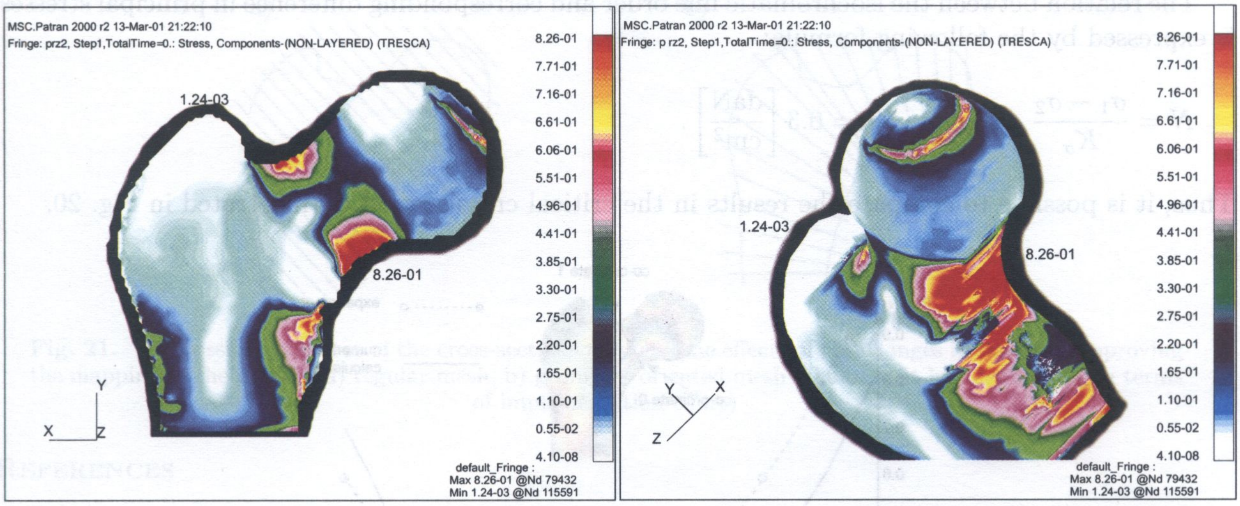

Fig. 17. Finite element analysis results (values relate to the model side surface)

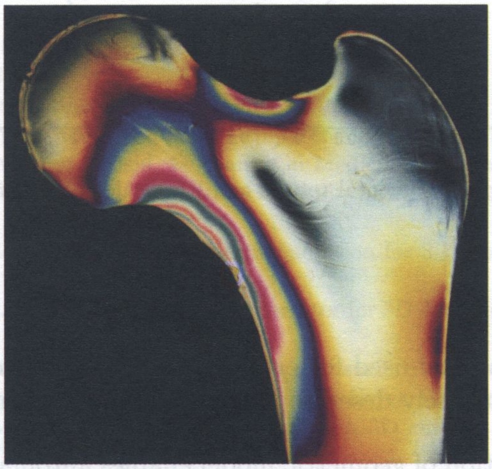

Fig. 18. Picture of photo-elastic image of the bone's longitudinal cross-section

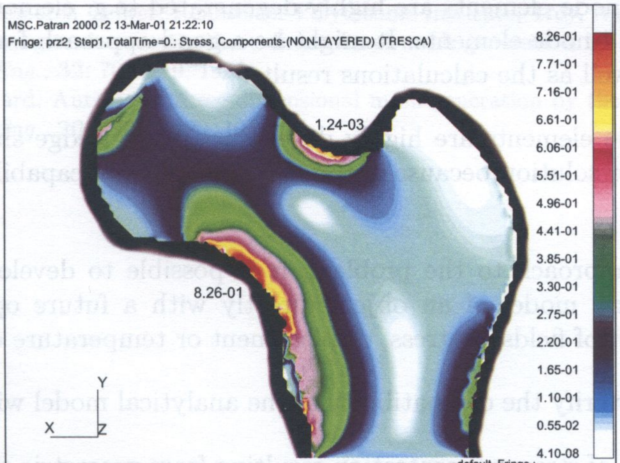

Fig. 19. FEA results (axial cross-section)

The relation between the isochromatic line order and corresponding difference in principal stresses is expressed by the following formula:

$$N = \frac{\sigma_1 - \sigma_2}{K_\sigma} \qquad \text{where} \quad K_0 = 0.3 \left[\frac{\text{daN}}{\text{cm}^2}\right].$$

Thus, it is possible to compare the results in the critical cross-section, as illustrated in Fig. 20.

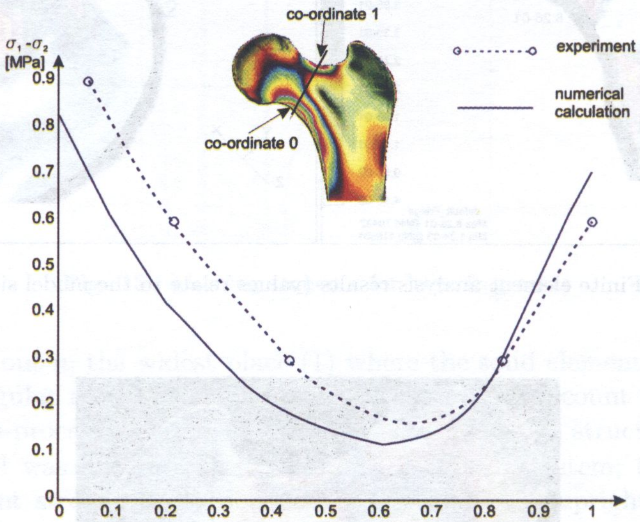

Fig. 20. Comparison of experimental results and FEA calculations in a selected cross-section

4. CONCLUSIONS

1. The proposed technique may be used for creating models of different shapes in situation where there is no possibility to precisely describe the geometry of the modelled object.

2. This technique of construction of geometric model enables automatic fitting over the model with a mesh of 8-node finite elements which offer better approximations of the field of displacement or stress than tetragonal elements.

3. In cases where the 8-node elements are highly degenerated (e.g. elements are wedge shaped), it is possible to use the 4-node elements. It might be a good approach for the sake of the meshing software abilities as well as the calculations results.

4. In cases where 8-node elements are highly degenerated (are wedge shaped), using 4-node elements might be better solution because of the meshing software capabilities and precision of the calculations results.

5. Due to algorithmic approach to the problem, it is possible to develop a suitable software to generate the geometric model of an object directly with a future option of automatic FEA calculations in respect of fields of stress, displacement or temperature distribution.

6. The results obtained verify the compatibility of the analytical model with the experimental one.

7. Unsatisfactory effects of stress concentration resulting from geometric step-like irregularities can be noted in the subsurface region. The influence of these steps can be reduced by making the division grid more dense, or by choosing the different positions of the intersection planes (Fig. 21).

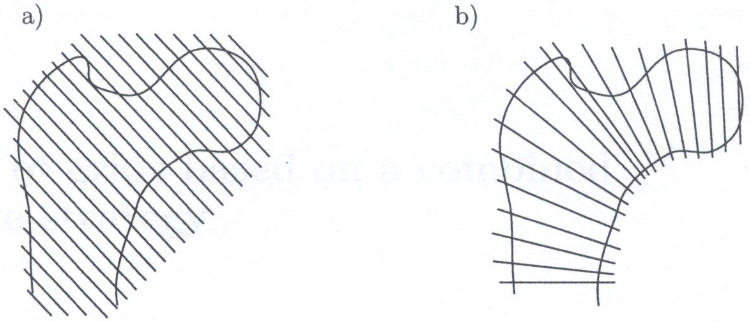

Fig. 21. The possible positions of the cross-sections reducing the effects of the changes in stiffness (improving the mapping of the model); a) regular mesh, b) geometry oriented mesh (the best fit but demanding in terms of implementation effort)

REFERENCES

[1] H. H. Barett Imagereconstruction and the solution of inverse problems in medical imaging. In. A. Todd-Pokropek and M. A. Viergever eds., Medical Images: Formation, Handling and Evaluation, NATO ASI Series, Springer, Berlin, 3–42, 1992.

[2] C. Barillot et al. Computers graphics in medicine: a survey, CRC critical rev. *Biomed. Eng.*, **15**(4): 269–307, 1988.

[3] H. Bieri, I. Metz. Algorithms for generalized digital images represented by bintrees. Technical Report IAM-91-001, University of Bern, 1991.

[4] R. Będziński. Engineering Biomechanics [in Polish]. Office of the Wrocław Technical University, Wrocław, 1997.

[5] J. Cwanek, H. Kopecki, M. Kopkowicz, A. Smykla. Computer-aided model analysing of disribution of internal forces acting in a hip joint element [in Polish]. Mechanics in Medicine, Sci. Seminar, Rzeszów, 2000.

[6] J. D. Foley, A. van Dam, S. K. Feiner, J. F. Hughes, R. L. Phillips. Introduction to computer grafics [in Polish]. WNT, Warszawa, 1995.

[7] P. Frey, B. Sarter, M. Gautherie. Fully automatic mesh generation for 3-D domains based upon voxel sets. *Int. J. Numer. Methods Eng.*, **37**: 2735–2753. 1994.

[8] P. L. George, F. Hecht, E. Saltel. Fully automatic mesh generator for 3D domains of any shape. *IMPACT Comput. Sci. Eng.*, **2**: 187–218, 1990.

[9] D. Karron, J. L. Cox, B. Mishra. The spider web algorithm for surface construction from medical volume data: geometric properties of its surface. *Technical Report*, New York University Medical Center, 1992.

[10] W. E. Lorensen, H. E. Cline. Marching cubes: a high-resolution 3D surface construction algorithm. *Comput. Graphics*, **21**(4): 163–169, 1987.

[11] MSC/PATRAN The PATRAN Command Language (PCL).

[12] B. van Rietbergen, H. Weinans, R. Huiskes, A. Odgaard. A new method to determine trabecular bone elastic properties and loading using micromechanical finite-element models. *J. Biomechanics*, Jan; **28**(1): 69–81, 1995.

[13] M. S. Shephard, M. K. Georges. Reliability of automatic 3-D mesh generation, SCOREC Report # 29–1991. Scientific Computation Research Center, Rennselaer Polytechnic Institute, Troy, NY, 1991.

[14] M. S. Shephard, M. K. Georges. Automatic three-dimensional mesh generation by the finite octree technique. *Int. J. Numer. Methods Eng.*, **32**: 709–749, 1991.

[15] M. A. Yerry, M. S. Shephard. Automatic three-dimensional mesh generation by the modified-octree technique. *Int. J. Numer. Methods Eng.*, **20**: 1965–1990, 1984.