11th INTERNATIONAL SCIENTIFIC CONFERENCE
NOVI SAD, SERBIA, SEPTEMBER 20-21, 2012

Stevanović, D., Korunović, N., Trajanović, M., Trifunović, M., Milovanović, J., Stojković, M.

FINITE ELEMENT MODEL OF HUMAN TIBIA AND PRELIMINARY ANALYSIS

Abstract: *A procedure for creation of subject specific finite element (FE) models of human tibia is described in the paper. Model geometry and material properties are based on medical images, obtained by computed tomography (CT) scanning. The most important steps of the procedure are: extraction of polygonal model from medical images, creation of surface model based on polygonal model and creation of FE model. An important step in subject specific FE model creation is assignment of material properties, which should also be subject specific. To achieve this goal, material properties were assigned to FE model by means of automatic mapping of CT numbers onto FE models, based on correlation between bone density and elastic properties of bone material. Optimal mesh density was established through convergence checks, performed on a series of FE tibia models of growing mesh density. The paper also describes a preliminary analysis of a created FE model, which was performed in order to check model integrity and validity.*
Key words: *Finite element method, reverse engineering tibia, computed tomography (CT)*

1. INTRODUCTION

Injuries or pathological processes (tumors and others) of human osteoarticular system are very common reasons for large number of surgical interventions. These interventions often require planning the process of surgery, as well as application of different implants or fixators. Nowadays, it is possible to use such medical aids due to increasing application of modern technologies in medicine, which enable computer-assisted planning of surgery [1]. Application of computer technologies results in accurate and precise geometry of bones and fixators [2]. The technique of Reverse Engineering (RE) is usually used for defining the precise geometry of bones [3], while the modeling of implants and fixators is performed by application of computer technologies such as Rapid Prototyping (RP) [4]. These techniques have been applied in planning of complex surgical interventions, modeling and constructing implants, medical devices, surgical tools, as well as in tissue engineering [5]. Mentioned technologies for precise defining of geometry may also be used for simulation of behavior of human osteoarticular system and nested implants or fixators, which would help achieve high efficiency in performing of certain orthopedic surgery.

Common bones fractures are those of long bones, and one of the most often is the fracture of tibia. Tibia fractures can be classified according to place, type and local mechanism of the fracture [6]. These fractures can be divided into two groups, that is, into fractures which appear at proximal and distal ends of tibia [7]. In order to enable easier treatment of the patient and optimize the shape of the implant or fixator which is to be used in the operation, it is necessary to apply the methods of simulation. Method of simulation based on finite element method was developed by Brekelmans in 1972, for the purpose of understanding mechanical behavior of the skeleton under mechanical loads in orthopedic surgery [8]. Brekelmans described the application of finite element method (FEM) in stress analysis of two-dimensional femur model. FEM based simulation of bone behavior and its application in orthopedics have been expanding ever since. Apart from the fact that this approach can provide answer to many scientific questions, there is still the need to improve this method for its easier and faster application in orthopedic surgery.

Large number of recent studies was based on general bio-mechanical behavior of the entire bone. The study described in this paper relies on bone material properties that are based on values of radiological tissue density, which were obtained from CT. Its aim was to create FEM model of tibia for stress analysis of tibia-implant or tibia-fixator assembly. It also included the creation of CAD model, necessary for FEM model construction. It is very important to note that accuracy of numeric solution of FEM model is influenced by following factors: boundary conditions, accuracy of the shape of finite element mesh created in CAD model based on CT scan data, and regularity of the finite elements shape.

2. PROCESS OF CREATING FEM MODEL OF TIBIA

Process of creating subject specific model of human tibia for testing the state of strain by application of FEM, is performed in following stages:

- Extracting and generating polygonal model based on CT scan.
- Creation of CAD model based on polygonal model.
- Creation of FEM model.

2.1 Extracting and generating polygonal model based on CT scan

First phase of extracting and generating polygonal model of tibia starts with collecting data obtained from computer tomography, that is, CT scan of lower extremities of the patient. In this case, resolution of CT image was 0.5 mm Fig. 1. [2, 3, 9].

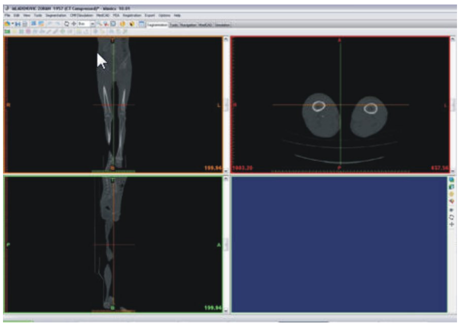

Fig. 1. Image of lower extremities obtained by computer tomography. Views in orthogonal coordinate system are presented in the figure

CT scan provides data necessary for creating "point cloud", which is later used in selected CAD software for the purpose of creating the geometrical image of the bone by reverse engineering. Using the applications for editing medical images obtained by computer tomography, interest zones for creation of FEM model are being selected. In this case, based on the selected zone, "point cloud" which corresponds to external surface of tibia and the surface which divides internal bone structure into area of bone tissue and area of bone marrow was created. During "point cloud" creation, a large number of points in internal bone structure that do not belong to the surfaces defining its internal zones are often selected Fig. 2. This problem is prominent in older patients, due to common appearance of osteoporosis. Selected points represent "noise" which poses problems in further generating of geometrical model [9].

Fig. 2. Polygonal model of tibia and surrounding bones parts, created based on "point cloud"

2.2 Creation of CAD model based on polygonal model

The following phase, after extracting and generating polygonal model, is removing parts and segments which do not belong to tibia. This primarily refers to removing parts of surrounding bones: femur, fibula and talus, as well as residual parts of soft tissue and veins Fig. 3., which leads to clean polygonal model for further creation of geometrical model.

After cleaning polygonal model, it is necessary to remove triangles on external bone surface which interfere in internal layer of the bone Fig. 4. Due to residual cracks in polygonal model it is also necessary to perform "healing".

Fig. 3. Polygonal model of tibia, with removed segments of surrounding bones and soft tissue

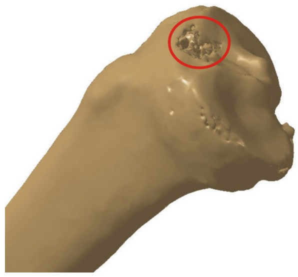

Fig. 4. Polygonal model with chaotic distribution of triangles, grouped according to internal structure of tibia. These cases are most often the consequence of osteoporotic bone scan

The following step is increasing the level of "smoothness", if necessary. Application of relevant approximation functions helps creation of NURBS surface which represents the external envelope of model of tibia Fig. 5. Final step, after forming the surface model, is creation of solid model of tibia, by filling the volume of external surface.

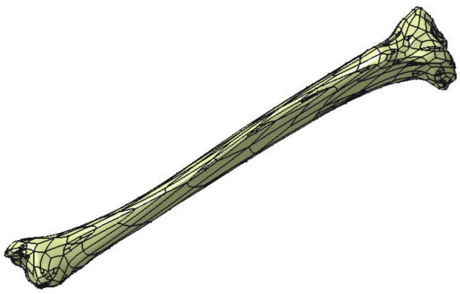

Fig. 5. NURBS surface which represents the external envelope of tibia

2.3 Creating FEM model of tibia

In this phase, the most important steps are creation of finite element mesh of optimal density and assigning appropriate material properties to the model. Process of material properties assignment can be performed in two manners. One of them is mapping the material that is, assigning characteristics of material to each finite element of the model, based on data obtained by CT scan of the patient. The other method is dividing internal bone structure into zones and assigning equivalent module of elasticity of bone tissue to corresponding bone zones.

2.3.1Creating FEM model for analysis of tibia

Finite element mesh on tibia model consists of tetrahedral finite elements of second order, the size of which grows towards the inside of the model.

The choice of optimal mesh density was based on results convergence test, where several consecutive meshes of growing density were used [10, 11]. Number of elements was increased according to the geometrical progression, whereas every side of the element formed in the previous step was divided by a half of its length.

The convergence was tested by comparison of changes in stress values between two consecutive solutions in selected areas Fig. 6., which are located in the zones with critical values of stress. The difference of stress values in selected nodes, compared to the previous step, was in the range from 0.057 % to 0.666 %. Since the difference was small, it was concluded that optimal mesh may be built using the average value of element edge equal to 2 mm and total number of elements is 145492. It is considered that the discretization error is reduced to minimum in this manner and that potential errors in analysis can be ascribed to other factors, such as material modeling or boundary conditions.

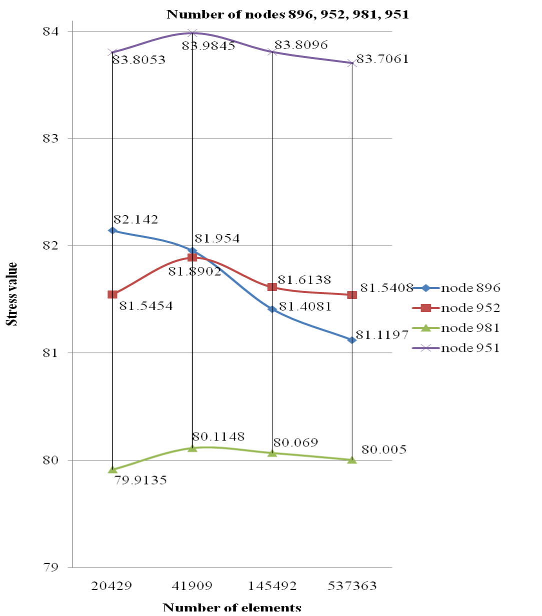

Fig. 6. Test of convergence of necessary mesh density based on monitoring changes of stress values in selected nodes

2.3.2Mapping of material on finite elements mesh

Material mapping, based on mechanical material properties obtained by correlation with CT image, represents the approach to material modeling which enables the fastest creation of the subject specific model of human bones. It is based on the fact that certain gray values on the CT scan, which are expressed in HU units (according to the Hounsfeld's scale), for known parameters of X-ray tube, correspond to certain values of radiological tissue density. On the other hand, these values can be correlated with apparent bone density. Different authors also analyzed the relation between density of different types of bones and their mechanical characteristics [12]. If it is assumed that the bone is non-homogenous and isotropic, the most common relation is the one between the module of elasticity and apparent bone density.

Morgan's relation [12] defined in 2003, was used in this study and it states:

$$E = 6950 \cdot \rho_{app}^{1,49} \cdot [MPa] \cdots \left(za\ \rho_{app}\ u \left[\frac{g}{cm^3} \right] \right) \quad (1)$$

where E represents the module of elasticity, and ρ_{app} is the apparent bone density. Obtained values of elasticity module are reduced to a hundred constant values; and based on these values, a hundred different materials were defined and assigned to finite elements of FEM model.

2.3.3 Description of preliminary analysis of created FEM model

Preliminary FEA of tibia model was performed, for the purpose of testing the integrity of the model and efficiency of practical application of described principles.

In the process of material mapping, it was necessary to take into consideration the thickness of compact bone. Since the bone is very thin on proximal and distal end, testing the convergence determines the density of mesh which best describes the structure of tibia in these areas, and the final result of the analysis contains the satisfactory level of accuracy.

Surfaces were created on the model envelope, which approximately correspond to the zones of joints and muscles activities. Loads taken from literature [7] were applied at proximal end of tibia, as equally distributed pressure acting on created surfaces of joints and muscles [13, 14]. Distal end of model was encastred.

Distribution of equivalent strain on the surface of FEM model of tibia, obtained by preliminary analysis, is presented on Fig. 7. Maximal values of the strain appear approximately at 1/3 of distance between distal and proximal ends.

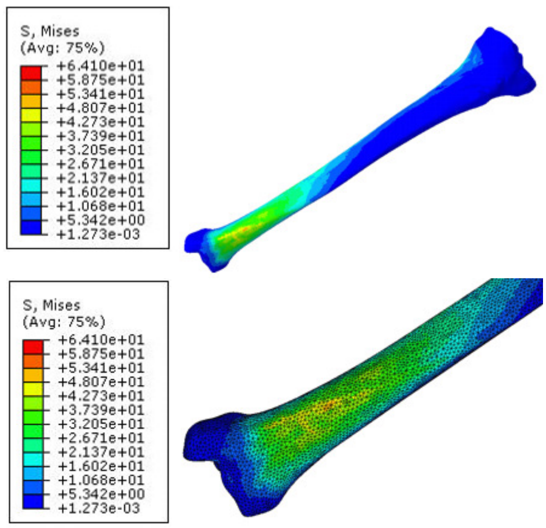

Fig. 7. Results of state of strain obtained by preliminary analysis as spotting critical points around distal end of tibia where the highest values of strain appear

3. CONCLUSION

Computer aided technologies have wide range of application in orthopedic surgery. This study presents their application in analysis of tibia, using FEM. Models obtained in this manner can significantly improve understanding of mechanical behavior of tibia exposed to different loading scenaria. Described procedure for creating FEM model of human tibia can also be used for creating models of other human bones.

Defined procedure for creation of FEM model enables subsequent solving of the following tasks:

- Creating model of bone-implant assembly and stress analysis of its components, for the purpose of choosing the material, optimal shape, dimensions and position of the implant.
- Detailed analysis of typical load cases on tibia, during everyday activities of the patient.

ACKNOWLEDGMENT

This paper is part of project III41017 Virtual human osteoarticular system and its application in preclinical and clinical practice, funded by the Ministry of Education and Science of Republic of Serbia, for the period of 2011-2014.

4. REFERENCES

[1] Trajanović, M., Mitković, M., Vitković, N., Milovanović, J.: *Definisanje zahteva aplikacije za planiranje operacija u hirurgiji koštano zglobnog sistema*, IMK-14 - Istraživanje i razvoj, Vol. 15(1-2), p.p. 5-11, 2009.

[2] Veselinovic, M., Vitkovic, N., Stevanovic, D., Trajanovic, M., Arsic, S., Milovanovic, J., Stojkovic, M.: *Study on creating human tibia geometrical models*, Proceedings of the 3rd International Conference on E-Health and Bioengineering - EHB, p.p. 195-198, Iaşi, Romania, Gr. T. Popa University of Medicine and Pharmacy Publishing House, Iaşi, Romania, 2011.

[3] Stojkovic M., Trajanovic M., Vitkovic N., Milovanovic J., Arsic, S., Mitkovic M.: *Referential geometrical entities for reverse modeling of geometry of femur*, Proceedings of the II ECCOMAS Thematic Conference on Computational Vision and Medical Image Processing: VipIMAGE 2009, p.p. 189-194, Porto, Portugal, Taylor & Francis Group, Porto, Portugal, 2009.

[4] Milovanović, J., Trajanović, M., Vitković, N., Stojković, M.: *Rapid prototyping tehnologije i materijali za izradu implantata*, IMK-14 - Istraživanje i razvoj, Vol. 15(1-2), p.p. 23-30, 2009.

[5] Simchia, A., Petzoldt, F., Pohl, H.: *On the development of direct metal laser sintering for rapid tooling*, Journal of Materials Processing Technology, Vol. 141 (3), p.p. 319-328, 2003.

[6] Christian, W., Peter, M., Leif, B. H., Tron, D. P. G.: *Finite element analysis of tibial fractures*, Danish Medical Journal, No. 5, 2010.

[7] Norio, S., Etsuo, C., Koji, T., Naoya, T.: *Biomechanical analysis for stress fractures of the anterior middle third of the tibia in athletes: nonlinear analysis using a three dimensional finite element method*, Journal of Orthop Science, Vol. 8 (4), p.p. 505-513, 2003.

[8] Brekelms, W. A. M., Poort, H. W., Slooff, T. J. J. H.: *A new method to analyse the mechanical behaviour of skeletal parts*, Acta Orthopaedica Scandinavica, Vol. 43 (5), p.p. 301-317, 1972.

[9] Korunović, N., Trajanović, M., Mitković, M., Vulović, S.: *Od CT snimka do modela za analizu naponskog stanja u femuru primenom metoda konačnih elemenata*, IMK-14 - Istraživanje i razvoj, Vol. 16(2), p.p. 45-48, 2010.

[10] Ingrassia, T., Mancuso, A., Ricotta, V.: *Design of a new tibial intramedullary nail*, Proceedings of IMProVe 2011 International conference on Innovative Methods in Product Design, p.p. 678-684, Venice, Italy, Università degli Studi di Padova Facoltà di Ingegneria Dipartimento di Architettura, Urbanistica e Rilevamento (DAUR), Venice, Italy, 2011.

[11] Viceconti, M., Davinelli, M., Taddei, F., Cappello, A.: *Automatic generation of accurate subject-specific bone finite element models to be used in clinical studies*, Journal of Biomechanics, Vol. 37, p.p. 1597-1605, 2004.

[12] Elise, F. M., Harun, H. B., Tony, M. K.: *Trabecular bone modulus-density relationships depend on anatomic site*, Journal of Biomechanical, Vol. 36, p.p. 897-904, 2003.

[13] Little, R. B., Wevers, H. W., Siu, D., Cooke, T. D. V.: *A three-dimensional finite element analysis of the upper tibia*, Journal of Biomechanical Engineering, Vol. 108, p.p. 111-119, 1986.

[14] Tarniţă, D., Popa, D., Tarniţă, D. N., Grecu, D.: *CAD method for three-dimensional model of the tibia bone and study of stresses using the finite element method*, Romanian Journal of Morphology and Embryology, Vol. 47 (2), p.p. 181-186, 2006.

Authors: **M.Sc. Dalibor Stevanović, Dr. Nikola Korunović, Prof. Dr. Miroslav Trajanović, M.Sc. Milan Trifunović, M.Sc. Jelena Milovanović, Dr. Miloš Stojković**, University of Nis, Faculty of Mechanical Engineering, Laboratory for Intelligent Production Systems, Aleksandra Medvedeva 14, 18000 Nis, Serbia, Phone.: +381 18 500-669.

E-mail: dalibor.stevanovic85@gmail.com
nikola.korunovic@masfak.ni.ac.rs
miroslav.trajanovic@masfak.ni.ac.rs
milant@masfak.ni.ac.rs
jeki@masfak.ni.ac.rs
miloss@masfak.ni.ac.rs