Jastrzębia Góra, 16th–20th September 2013

YOUNG'S MODULUS DISTRIBUTION IN THE FEM MODELS OF BONE TISSUE

Katarzyna Mazur[1] and Leszek Dąbrowski[2]

Faculty of Mechanical Engineering, Gdańsk University of Technology,
ul. G. Narutowicza 11/12, 80-233 Gdańsk, Poland
[1]katmazur@student.pg.gda.pl, [2]ldabrows@pg.gda.pl

ABSTRACT

This paper presents how differences of Young's modulus in adjacent finite elements typical for organic materials such as bone tissue, influence stress calculating. Emphasizing high computational cost of variable Young's modulus in parts of the model, where the number of finite elements has been raised, the authors wants to prove that new model of finite element which has variable Young's modulus in its volume needs to be created. The article contains a description of the computer simulation and the interpretation of their result.

INTRODUCTION

Finite element method (FEM) allows precise calculations of the strength of objects of known and regular materials properties. Typical FEM calculations in mechanical engineering describe models made of materials of uniform stiffness. Stress concentration revealed on the border between the uniform areas of different stiffness is a real and well–known phenomenon. Repeatability and uniformity of properties such as Young's modulus, however, do not apply to organic materials, such as wood or bone.

Image of local tissue stiffness variation in one bone area is available through Computer Tomography (CT). In the biomechanics literature, many publications on testing and verification FEM models created from CT images are available [1, 2], while in some other publications their authors aim to draw medical conclusions from such calculations [3]. The authors of [1, 2] experimentally confirm the quality of bone tissue FEM models that has been built based on the relation between Young's modulus and radiographic density (expressed in Hausfield's units) using an intermediate relation of mineral density ρ and radiological HU

$$\rho\left[\frac{\mathrm{g}}{\mathrm{cm}^3}\right] = \frac{0.63HU - 6.7}{1000}\,, \tag{1}$$

$$E[\mathrm{MPa}] = 1904\rho^{1.64}\,. \tag{2}$$

Instead of using Eq. (1) sometimes CT technique called quantitative computed tomography QCT is used.

This paper presents inaccurate results in currently used meshing algorithm for FEM of bone tissue and proposes a solution to this problem. The simulations are a part of a doctoral thesis undertaken at the Faculty of Mechanical Engineering at Gdańsk University of Technology.

VARIATION OF YOUNG'S MODULUS IN THE NEIGHBORING FINITE ELEMENTS OF BONE TISSUE MODEL

The image of a CT scan has a limited resolution like an image of digital photography. An elementary part of such image is called a vortex. Construction of the FEM model of a bone from CT images in such a way that one vortex is represented by one finite element is the best way to use data obtained from CT. Using smaller number of finite elements neglects some part of information of the variable stiffness of the material within the bone. The use of a larger number of elements requires interpolation of material stiffness value, which decreases the quality of the information, and increases the size of the FEM problem.

In Fig. 1 we can see the steps of such a process. Tomographic cross-section of the femur with applied tomographic density scale (Fig. 1(b)) at the level marked in Fig. 1(a) has been transformed into the FEM model (Fig. 1(d)) with assigning Young's modulus according to (1) and (2) to each of the elements (rating effort tissue is generally made on the basis of von Misses stress [4]). Radiation density distribution on femur section (indicated in Fig. 1(b) by r–axis) is shown in Fig. 1(c). This is an usual distribution (with the maximum in the middle of the wall thickness) and it is continuous, due to interpolation of measurement data.

The calculation of stress distribution (Fig. 1(d)) has a clearly discontinuous character. Very significant stress gradients appear on finite elements' boundaries that can only be explained by the imperfection of the calculation method. This imperfection in FEM is ignored in publications of bone modeling and will be examined later in the work.

EFFECT OF VARIATION OF YOUNG'S MODULUS IN THE NEIGHBORING FINITE ELEMENTS

For the needs of the simulation, a virtual sample size 50 x 50 mm was created, built of 25 CT vortices (in a grid of 5 x 5). The sample is supported on the lower edge. The upper edge is displaced by u_y (Fig. 2) in such a way that the compressive stress amounts to 100 MPa. Continuous and smooth (linear) distribution of material properties (Young's modulus) was applied in the calculated area to make it easy to detect any anomalies in the calculated stress distribution

$$u_y = \frac{h\sigma}{E} = \frac{50 \times 100}{2.1 \times 10^5} \left[\frac{\mathrm{mm} \times \mathrm{MPa}}{\mathrm{MPa}}\right] = 0.024[\mathrm{mm}] \ . \tag{3}$$

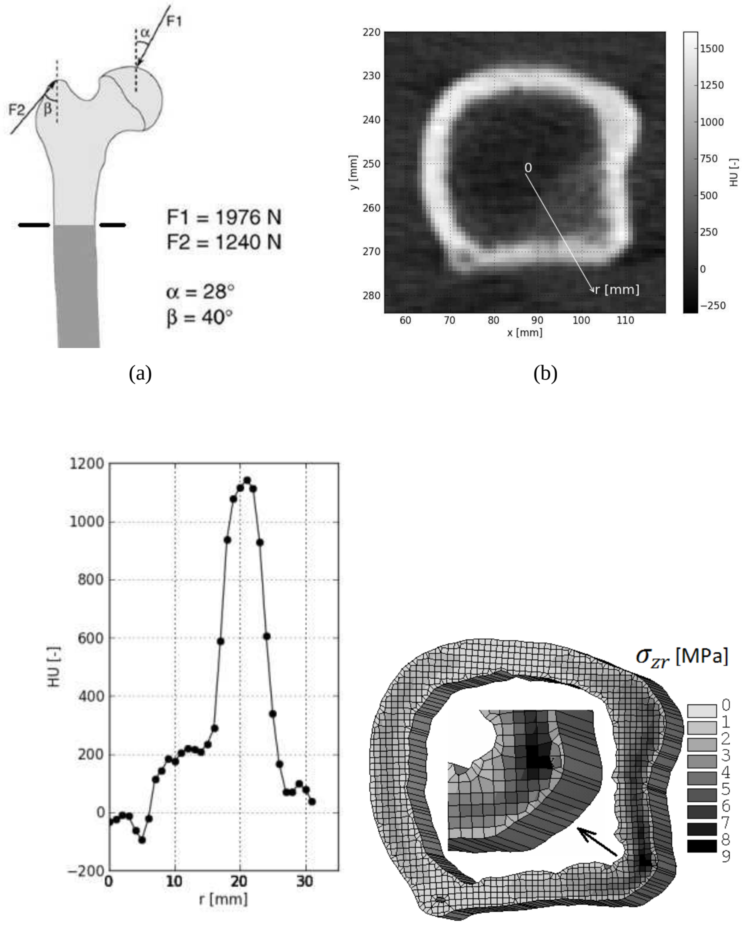
According to calculations illustrated by Fig. 1 and explained below this figure, each of vortices was converted into the finite element and received its own Young's modulus. Modulus' distribution defined by

$$E(x, y) = ax + by + c \ . \tag{4}$$

Divided into vortices coefficients a, b and c were chosen so that in the sample's corners' (described by the coordinates according to Fig. 2) Young's modulus the following values $E(0,0) = 1[\mathrm{Pa}]$, $E(0,50) = E(50,0) = E_s = 2.1 \times 10^5[\mathrm{MPa}]$ and $E(50,50) = 2 \times E_s$. The assumption of constant value of Young's modulus in the element's area caused that the spatial distribution of the module is in the form of a stepped pyramid (Fig. 3).

Two subsequent simulations were carried out for meshed the sample with the use of ANSYS program. The first simulation was done in accordance to the principle described above: one finite element ⇒ one vortex. In the second simulation, finite element mesh was refined so that each vortex was represented by 400 finite elements (20 × 20), but the distribution of the Young's modulus remained unchanged (*i.e.* according to Fig. 3). The results of both simulations are presented in Fig. 4.

In the results of the first simulation (Fig. 4(a)) one can observe that stress contour boundaries coincide with the boundaries of finite elements. They are sharp and irregular, which is impossible



(a) (b)

(c) (d)

Figure 1. (a) Diagram of the femur load. (b) Femur tomography image with radiological density scale applied. (c) Radiological density distribution along the marked radius. (d) Equivalent stresses by von Misses calculated FEM with finite elements based on vortexes Fig. (b). Presented results are authors' own research.

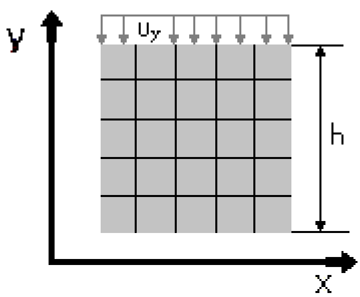


Figure 2. Meshed virtual sample and its displacement.

for the assumed distribution of Young's modulus. A slight improvement of the stress concentration can be observed in the results of the second simulation (Fig. 4(b))—when meshing was finer. This

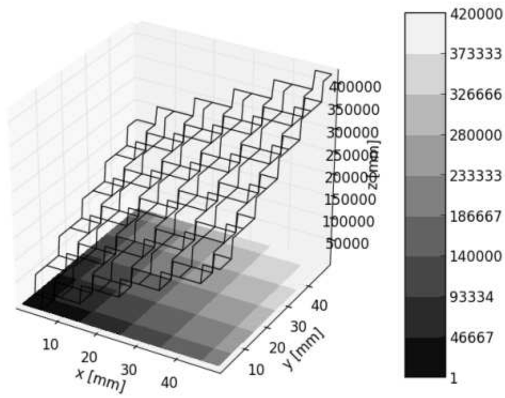


Figure 3. Graph of the Young's modulus E [MPa] in the 25 finite elements coordinates.

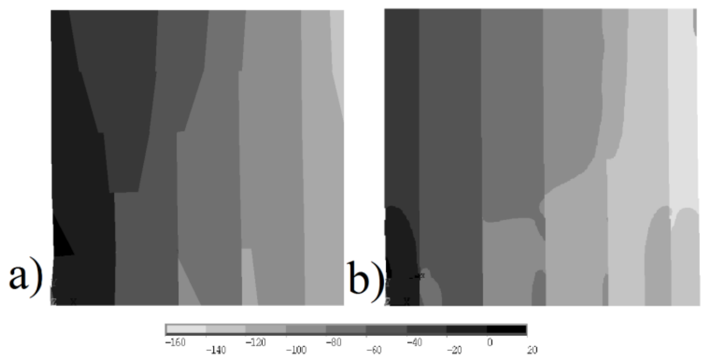


Figure 4. Interpolated nodal values of calculated compressive stress σ_y [MPa]: (a) sample with an aspect ratio 1 vortex = 1 finite element, (b) sample with an aspect ratio 1 vortex = $20 \times 20 = 400$ finite elements.

procedure however resulted in a significant increase of the calculations' time. The contour lines clearly coincide with the boundaries of finite elements, but they are smoother, which gives the impression of smoother transition between successive stress values. The modeled phenomenon has a completely continuous spatial distribution, so the apparent lack of continuity in the results seems to be an error of the used method, because distribution of the elements in the studied area is an artificial meshing, in fact there is no boundaries, so we recognize as an error if any boundaries appears in calculations results).

THE EFFECT OF MESH REFINEMENT WITHOUT CHANGE OF YOUNG MODULUS DISTRIBUTION

The examples described in the previous paragraph show that an increase of the number of elements, while the distribution of Young's modulus remained unchanged improved the continuity of the results only slightly. In subsequent numerical experiments, a fine mesh of 100 $\times$ 100 = 10 000 elements was used. An individual Young's modulus value is assigned to each of these elements according to the equation (4)—Fig. 5.

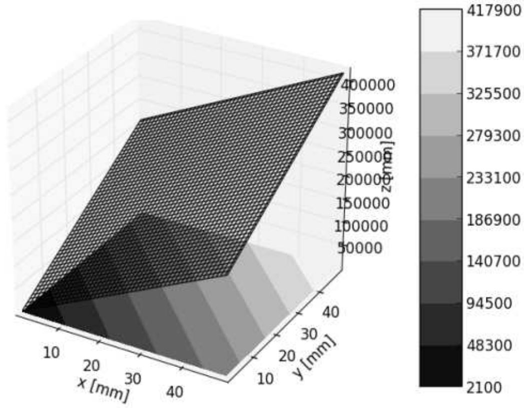


Figure 5. Graph of Young's modulus as a function of the coordinates of 10 000 finite elements.

As expected, the change in the method assigning values of Young's modulus to the finite elements caused a significant change in the obtained results (Fig. 6). Contour lines are no longer closely associated with the vortices boundaries; they are significantly smoother than either of the previous results. Computational time in this case was similar to the second simulation case (Fig. 4(b)).

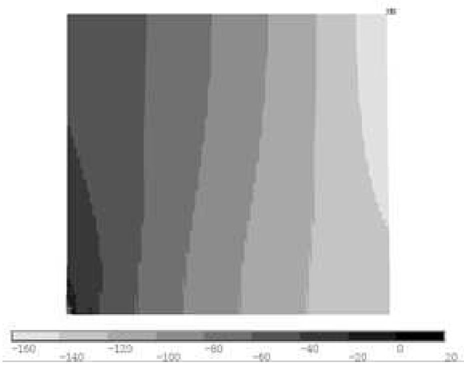


Figure 6. Calculated compressive stress σ_y [MPa] in the sample division on 10 000 finite elements, each with its individual Young's modulus value.

CONCLUSIONS

The use of mesh density usually described in the biomechanics literature leads to large discontinuity of calculated stresses. Refining the mesh and maintaining the spatial distribution of Young's modulus (as derived from CT density of radiation) does not make a significant improvement, but significantly increases the computational time.

Clear improvement of the results can be obtained by using fine mesh division and performing interpolation of Young's modulus and assigning its individual value to each finite element. Unfortunately, this approach increases the computational time beyond the level acceptable in medical diagnostics. The solution for future, according to the authors is to develop a new type of finite element, in which material stiffness would not be constant variation in its material stiffness. For example this can be done by assigning a different value of Young's modulus to each node of the finite element and perform interpolation of the value within the element volume. Such an approach has not been proposed before and will be the subject of authors' future research.

REFERENCES

[1] L. Grassi, E. Schileo, F. Taddei, L. Zani, M. Juszczyk, L. Cristofolini, and M. Viceconti: *Accuracy of finite element predictions in sideways load configurations for the proximal human femur*, J. Biomech. **45** (2012), 2–3.

[2] R. Fedida, Z. Yosibash, Ch. Milgrom, and L. Joskowicz: *Femur mechanical stimulation using high–order FE analysis with continuous mechanical properties*, II International Conference on Computational Bioengineering (2005), 5–9.

[3] T.N. Gardner, T. Stoll, L. Marks, S. Mishra, and M. Knothe Tate: *The influence of mechanical stimulus on the pattern of tissue differentiation in a long bone fracture—an FEM study*, J. Biomech. **33** (2000), 415–425.

[4] A. Boryor, M. Geiger, A. Hohmann, A. Wunderlich, Ch. Sander, F. Martin Sander, and F.G. Sander: *Stress distribution and displacement analysis during an intermaxillary disjunction—A three–dimensional FEM study of a human skull*, J. Biomech. **41** (2008), 376–382.