Finite element simulations of stresses in bone implants made by three-dimensional printing

V M Mikushev¹, A I Samarkin² and A S Khomutova¹

¹ Department of Solid State Physics, St Petersburg State University, 7/9 Universitetskaya Emb., St Petersburg, 199034, Russia
² Department of Medical Informatics and Cybernetics, Pskov State University, 2 Lenin Square, Pskov, 180000, Russia
E-mail: v.mikushev@spbu.ru

Abstract. The article is devoted to the analysis of the strength properties of a compact bone tissue in order to produce subsequently the equivalent in strength individual implant. An example of a fragment of a skeleton bone shows the fundamental possibility of solving the problem using a 3d modelling finite element analysis software package.

1. Introduction
In 2013, 1518 patients with bone tumors were identified in Russia [1]. At the same time, in almost sixty percent of cases, they are localized in the extremities, a greater number of such patients are shown crippling treatment as an operation. In the USA, 8680 only new cases of CMT are diagnosed annually, in 60% of cases these tumors are localized in the extremities [2]. Every year in the world, about 3100 people die from malignant tumors of soft tissues, the latency during the first year after diagnosis is 32.0-34.5% [3]. Due to the low efficiency of chemoradiation treatment, the main method of treatment is surgical [4]. It consists of wide excision of the tumor in a single block with all the vessels, which allows removing the tumor in full (see figure 1).

Figure. 1 Examples of bone tumors. Arrows indicate the localization of the distal focus of the metastatic bone.
To avoid amputation requires a complex reconstructive stage using synthetic prostheses [5], preferably made for each specific patient. At the same time, the structure of bone tissue is very complex both structurally and in terms of its genesis, including the effect of neoplasms on its strength.

Thus, a number of problems arise, mainly of a technical nature, namely: modelling the bone tissue and its strength properties, assessing the stresses between the bone and the installed implant, achieving a favorable character of these stresses for the patient.

2. Methods

2.1. About bone tissue modeling by the finite element method

The solution of complex problems associated with the strength calculations of individual elements and their systems is currently carried out almost exclusively by finite element methods (FEA). At the same time, modelling of materials with a complex structure (composites, lattice structures by FEA methods is difficult, due to the high computational costs of calculating the interaction between structural elements.

In this regard, we are forced to consider the issue of the structure of bone tissue and methods of its modelling. 80% from the mass of all the bones of the body forms a compact bone tissue [6], in the structure of which osteons are allocated [7]. This structural element consists of cylinders consisting of bone plates inserted into each other. The number of plates varies from 3 to 25 [8]. The thickness of the bone plate is 4-15 microns. Each bone plate contains a large number of osteocytes, which are located in the bone lacunae. The osteocyte has a stellate shape with branching processes. The diameter of the osteon is 0.3-0.4 mm. The Havers canal is located in the centre of the osteon, in which two small blood vessels pass. The diameter of one capillary is 6-20 microns, and in channel n there are two capillaries, so the diameter of the Havers channel can be taken as 40 μm. [8, 9]

![Figure. 2 Compact bone tissue structure (from [7]).](image)

For modelling bone tissue, it is proposed to use the same approach that the authors used in [10] to analyze lattice structures in three-dimensional printing, namely, to simulate a single osteon (with or without osteocytes modelling). Based on the data obtained, calculate the averaged characteristics and then use them in the calculations. Despite some simplifications, the results of such modelling are in good agreement with the more complex techniques described, for example, in [11]

2.2. Bone and implant modeling methods

The peculiarities of the patient's body involve the use of personalized bone models and the individual construction of the affected area to be replaced by the implant. These tasks can be solved by importing the results of CT and MRI scans into the modelling environment. The presented models are created in Stereo Lithography (STL) format. Despite the known shortcomings of the format, it is quite common, exported and readable by most simulators, and relatively easy to fix.
Certain difficulties are caused by the modelling of neoplasms and their effect on the strength of the surrounding bone tissue. Further, it is assumed that the removed part of the neoplasm completely covers the weakened zone.

2.3. Increased compliance of the implant
Implant design is also an important issue. Despite the presence of examples of the implementation of self-dissolved constructions [12], at present, bone replacement elements made of titanium are more often used. In this case, difficulties arise with the manufacture of implants with a geometry unique for each patient and a problem with contact stresses in the area of the conjugation of the bone tissue with the titanium plate, since the strength of titanium alloys is significantly higher than the strength of bone tissue.

Customized implants can be produced using 3D printing methods. To reduce contact deformations, lattice implants should be made. The degree of attenuation can be calculated by analyzing a regular lattice structure or by topological optimization of the implant design.

3. Modeling

3.1. Simulation of single osteon
To simulate a single osteon, it is proposed to use the parametric model developed by the authors and presented below. The model uses the idea of bone plates as shells. Thus, the parameters of the model are the number of shells and the distance between them. The improved model includes lacunae that simulate osteocytes. Osteocytes are modelled as a parameterized 3D spiral array, as shown in figure 3,

The computational model provides for the creation of a mesh of finite elements, for which an automatic refinement of the size around the osteocytes is provided (refinement factor 2).

Let us set the following values of the initial parameters for carrying out model strength calculations:
1. Length of the model bone fragment (cylinder) \( L = 0.4 \text{ mm} \); 2. Fragment diameter \( D = 0.4 \text{ mm} \); 3. Axial load \( F = 0.2 \text{ H} \). Note that in both models of bone tissue, the deformation values are practically the same - 6.49 μm and 6.33 μm, and the equivalent stresses for the model with osteocytes are \( 4.2 / 2.8 = 1.33 \) times higher, with the localization area stress is the zone of fixation of the base in the area of its intersection with the osteoblast cavity, as shown in figure 4.

Figure. 3 Shell-based parametric osteon model (a – load scheme, b – simple shell model, c – shell model with 3d array of osteocytes).
Thus, under the accepted assumptions, we can assume that the compact bone substance has the following characteristics (Table 1):

Table 1. Mechanical parameters of compact bones tissue.

| N | Parameter       | Value    |
|---|----------------|----------|
| 1 | Young’s module | 98 MPa   |
| 2 | Density        | 2100 kg/m³ |
| 3 | Poisson’s module | 0.33     |

3.2. Simulation of the behavior of healthy and tumor-affected bone under stress

The presented results of three-dimensional scanning of skeleton bones by computed tomography methods serve as a bone model. Note that the directly available scan results need serious revision by averaging coordinates, excluding small polygons and stitching cavities with patch polygons to achieve a closed bone geometry. Modern CAE analysis environments have automatic and semi-automatic tools for correcting raw geometry. Below in figure 5 the finite element mesh of the corrected healthy bone model (a) and visually simplified version with applied boundary conditions (b) is presented.

Figure 4 Von Mises equivalent stresses (a, c) and deformation (b, d), where a and b is solid-shell model, c and d – with osteoblasts.
As the maximum load, consider the force acting on the bone when the patient jumps on one leg. Let the patient's mass be \( m = 120 \text{ Kg} \), the dynamic coefficient \( k_d = 1.7 \) (taking into account the damping properties of tissues and cartilage of the skeleton), \( g = 9.81 \text{ m/s}^2 \), then the load can be estimated in \( F = k_d mg = 2000 \text{ N} \) and is directed along with the axis bones (horizontally in the picture above).

Neglecting the effect on the strength of the cancellous part of the bone, one can set the parameters of compact bone tissue discussed above (see Table 1).

We use a sequence of calculations typical for CAE systems: import of geometry and its correction, setting material parameters, creating a calculation scheme and determining boundary conditions, building a finite element mesh, numerically solving the resulting model and post-processing the results.

Figure 6 shows the calculation results: von Mises stress diagram, original mesh, deformed bone with a zone of increased stress.

![Figure 6](image)

**Figure. 6** Calculated von Mises stress in whole bone

With the previously adopted bone strength limit of 160 MPa, we obtain a safety factor of about 25%, which corresponds, for example, to the information given in [7].

Consider the influence of tumors on bone strength. Suppose the affected area is ball-shaped and completely removes bone. In this case, the position of the sphere and its size can be specified parametrically (see in figure 7a sample of tumor sphere and in figure 7 b sample of results).

Modelling was performed with bone strength three positions tumors (in the upper, middle and lower parts) while varying the radius of the sphere of the tumor, as shown below in table 2.

| Bone part          | Radius of tumor sphere, mm |
|--------------------|----------------------------|
|                    | 5  | 10 | 15 |
| Near the knee      | 129| 129| 132|
| Middle part of the bone | 355| 267| 400|
| Near the neck of the thigh | 347| 542| 880|
Calculations show that with damage to the bone tissue near the knee joint (at a distance of about 100 mm along the axis of the bone), the maximum stresses increase insignificantly up to 129 MPa, however, in the middle part of the bone and near the femoral neck, even an insignificant neoplasm causes terminal consequences for the bone. Let us pay attention to a decrease in stresses in the middle part of the bone with an increase in the radius of the sphere to 10 mm. This is due to the concentration of stresses on cavities with a small radius.

Consider a “sparing” implant design that replaces the removed bone tissue. In view of the limited volume of the article, we will limit ourselves here only to the localization of the implant in the middle part of the bone (see in figure 8).

The maximum stresses of 290 MPa are observed in the contact zone of the plate with the bone along the loading axis and are of a local nature (already at a distance of 3-5 mm they do not exceed 75 MPa). However, this phenomenon can potentially lead to bone damage. These undesirable phenomena can be eliminated by increasing the compliance of the implant to values close to the bone material. In this work, an example of an implant is given, the increased stiffness of which is achieved due to the holes. Nevertheless, there is a more promising way - the manufacture of a lattice structure implant using the 3D printing method.
When using an implant, the flexibility of the material of which is increased due to the presence of holes, it is possible to redistribute stresses evenly over the entire material of the implant plate. In this regard, the stresses in the implant increase, but the contact stresses on the transition surface from the implant to the bone are reduced to 64 MPa on average, and local stresses are reduced to 103 MPa.

4. Conclusions
Thus, the use of three-dimensional modeling technologies, individual manufacturing of prostheses of affected areas of bone tissue using three-dimensional printing methods reduce bone stress and can allow patients to lead a better lifestyle.
Measures to increase the compliance of the implant allow a more even distribution of stresses in the bone-implant pair, which reduces the risk of bone destruction in the area of contact with the prosthesis.

5. References
[1] Kononovich N A, Popkov A V, Popkov D A and Shastov A L 2018 Scientific notes Kazan Bauman state academy of veterinary medicine 236 105–11
[2] Van der Graaf W T A et al 2012 The Lancet 379 1879–86
[3] Ek E W, Rozen W M, Ek E T and Rudiger H A 2011 Archives of orthopaedic and trauma surgery 131 487–95
[4] Bielack S et al 1999 Klinische Padiatrie 211 260–70
[5] Wei F-Q, Sun W, Wong T-S, Gao W, Wen Y-H, Wei J-W, Wei Y and Wen W-P 2016 Journal of experimental & clinical cancer research : CR 35 18
[6] Burt L A, Greene D A and Naughton G A 2017 Pediatric exercise science 29 456–64
[7] Bilezikian J P, Martin T J, Clemens T L and Rosen C J (eds) 2020 Principles of bone biology (London, United Kingdom: Academic Press, an imprint of Elsevier)
[8] Netter F H 2014 Atlas of Human Anatomy, Professional Edition: Including NetterReference.com Access with Full Downloadable Image Bank (Netter Basic Science) 6th edn (Saint Louis: Elsevier Health Sciences)
[9] Drake R L, Vogl W, Mitchell A W M and Horn A 2020 Gray’s anatomy for students (Philadelphia, PA: Elsevier)
[10] Samarkin A I, Samarkina E I, Mikushev V M and Plohoі I V 2019 AIP Conference Proceedings 2188 40004
[11] Narayan R (ed) 2020 Rapid prototyping of biomaterials: Techniques in additive manufacturing (Woodhead Publishing series in biomaterials) (Duxford: Woodhead Publishing, an imprint of Elsevier)