Algorithm of generation finite element mesh for the system «vertebrae – intervertebral disk – vertebrae» based on the stl model

Konstantin Kurochka, Kosntantin Panarin, Ekaterina Karabchikova
Gomel State Technical University named after P.Sukhoj
Gomel, Belarus
kurochka@gstu.by, logran2@gmail.com, catharine-diva@gstu.by

***Abstract*—The authors propose an algorithm for generating a regular finite-element mesh, based on the three-dimensional «vertebrae-intervertebral disk-vertebrae» stl model, that resistant to defects in the model, which allows the model to be correctly partitioned if there are intersecting objects or objects with incompletely touching surfaces. The resulting finite elements of different objects will be guaranteed to match, that is, the sides and edges of neighboring elements will be guaranteed to repeat each other.**

***Keywords*—Mesh, finite elements, stress-strain state, modeling, vertebra, stl**

I. Introduction

In the study of the state of the lumbar spine of a person, very often there is the problem of identifying areas of the spine that are exposed to the greatest loads. This will allow to choose the best treatment method and, as a result, minimize the rehabilitation period. In addition, when choosing a surgical procedure, it is often necessary to estimate how the stress-strain state of the vertebra changes, how the surgery affects the stress-strain state of other vertebrae, and how the stress-strain state of the lumbar spine changes over time after surgery [1], [2].

To estimate the state of the lumbar spine usually used radiographic methods, which make it possible to get a visual picture of the current state of the spine [3] (Fig. 1).

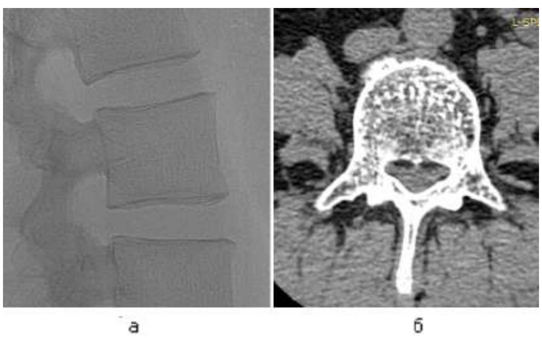

Figure 1. An example of radiography (a) and computer tomography (b) images

But radiographic pictures do not allow us to estimate the stress-strain state or predict the development of the resulting stresses over time.

Thus, when treating a patient and examining the state of his lumbar spine, necessary to solve a number of problems presented in Fig. 2

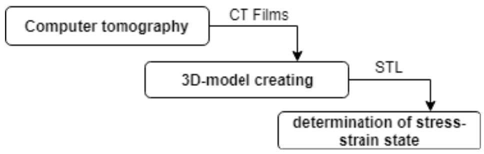

Figure 2. The general scheme of studying the state of the lumbar spine

To generate STL files based on the results of CT images obtained in the DICOM format, an algorithm based on vertebral search and localization [4] was used.

To determine the stress-strain state of mechanical and biomechanical systems, in practice very often the finite element method [5] is used, which allows one to find deformations from the action of different loads on any complex non-uniform three-dimensional objects.

II. Existing solutions to the problem

When using the finite element method, the initial stage is the approximation of the studied area and its discretization into grid cells (finite elements). This stage is one of the key and most significant ones, as further calculations are completely based on the obtained results of discretization the source domain. Due to the various features of the structure, shape and size of the studied three-dimensional models, an individual approach is required for the correct execution of the discretization stage, except for cases with simple models. Finite element meshes can be separated into regular (fig. 3 a), containing objects of the same shape and size, and irregular (fig. 3 b), varying the size of the elements for maximum match the original shape [6].

When discretizating objects of complex shape on an irregular grid, there may be places where finite elements acquire an irregular shape, an uneven size of the sides

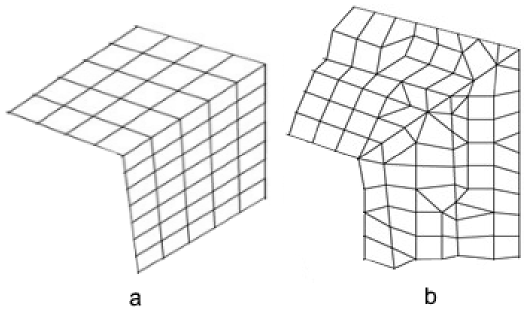

Figure 3. Regular (a) and non-regular (b) finite element mesh

(the size of one side greatly exceeds the others) or the finite element becomes too small. As a consequence of these problems, the calculation becomes impossible, or the accuracy of the calculation results based on a such model decreases.

In this paper, the model is a «vertebrae – intervertebral disk – vertebrae» system, which is determined by a stl-file with a three-dimensional model (Fig. 4) describing the surfaces of vertebrae and intervertebral disks.

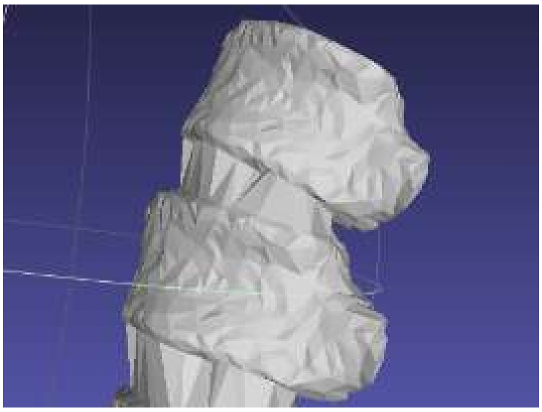

Figure 4. Example of stl-model «vertebrae - intervertebral disc - vertebrae»

On the basis of this model, the construction of a finite element mesh is carried out. This file was formed as a result of reconstruction of the spinal column according to the results of image analysis obtained using computed tomography and X-ray, in connection with which the objects of the stl-model have uneven edges, and may not completely touch the surfaces or intersect and overlap. Described factors impose limitations for partitioning the model into finite elements by standard means.

Existing software for mesh creating, such as NetGen [7] and TetGen [8], allows to customize various parameters of a finite element mesh, but when constructing a mesh, they produce a discretization based on their algorithms not only on the internal area of a three-dimensional object, but also on its surface. The result of approximation of the surface by finite elements is its simplifications and distortion, in consequence of which between the vertebrae and intervertebral discs are formed the intersection of surfaces or gaps with the lack of direct contact, which results the impossibility of further calculations.

Using Gmsh software [9] allows you to avoid this problem, since this software performs the discretization of the internal area of the object based on the existing surface points. The resulting finite element meshes have the original correctly touching surfaces, without gaps and intersections. However, Gmsh makes it impossible to correctly split into finite elements wide and flat objects, such as an intervertebral disk. With a small distance between the upper and lower planes, the application performs a separation by connecting the points of these planes directly, which leads to the appearance of a small number of finite elements whose dimensions of the sides are comparable to the size of the entire disk. The presence of such elements makes it impossible to correctly calculate the stress-strain state of the spine.

III. Defect-resistant regular mesh partitioning algorithm

The authors propose an algorithm for constructing a regular finite-element mesh based on a three-dimensional model of the human spine, that allow to avoid intersections of the boundaries of elements of various objects, resistant to arbitrary irregularities of the surfaces of model objects, defects of objects, as well as gaps between objects. The application of the proposed algorithm to the stl-file allows you to get a regular finite element mesh for the entire model.

The algorithm for constructing a finite element mesh can be separated into the following steps:

- the establishment of maximum and minimum values of the length, width and height of the analyzed model and the formation of the corresponding region of space;
- dividing the resulting region of space into a finite set of regular parallelepiped cells by forming vertical and horizontal lines — constructing a regular grid;
- finding the set of points of intersection of the formed straight lines with the elements of the objects of the analyzed three-dimensional model;
- removal of parallelepiped cells that are not related to any of the objects contained in the model;
- the division of parallelepipeds into finite elements (tetrahedra);
- correlation of the resulting tetrahedra with the corresponding model objects.

The first step of the algorithm is to establish a working region of space in which the model is considered. This area is a parallelepiped wrapping the model, to obtain the dimensions of which, the points (vertices) of the considered model are searched for, which have the smallest and largest values for each of the coordinate axes.

The second stage of the algorithm is based on the concept of regular (structured) meshes, according to which the model under consideration is completely placed in the resulting region, on which the grid is subsequently superimposed. This mesh is formed by constructing a set of straight lines defined by a fixed step in three directions of coordinate axes. In this case, the step in each of the three directions may differ from each other, which allows to vary the shape and size of the final finite elements.

In other words, at this stage determined by a plurality of straight lines that define a set of ribs parallelepipeds serving as the basis for the further formation of the finite element.

At the third stage, the whole set of triangular elements included in the three-dimensional stl-model presented is traversed in order to find the points of intersection with the previously constructed set of lines forming a regular mesh. The points of intersection of lines with triangles obtained as a result of this stage are fixed, thereby forming a set of segments located inside the region defined by the initial stl-model.

As a result of the analysis of the obtained segments in the fourth stage, a set of parallelepipeds is compiled, which are guaranteed to be part of the three-dimensional model under consideration. Parallelepipeds, in the construction of which the obtained segments were not involved, are discarded.

Then each parallelepiped is discretized into 6 equal tetrahedra. For each tetrahedron checked affiliation of its model and determines which element of the lumbar spine it belongs. Depending on the element belonging, its physical and mechanical characteristics (elastic modulus and Poisson's ratio) are determined.

A total of 4 different structural elements are considered [10]:

- the cortical tissue;
- the intervertebral disk;
- the spongy tissue;
- the transverse processes.

An example of a finished finite element mesh is shown in Fig. 5

With such partition into finite elements boundary planes of different objects are guaranteed to be the same, i.e. the part of parallelepipeds forming neighboring finite elements and their eedges obtained as a result of sampling will be guaranteed to repeat each other, which would eliminate the possibility of the empty space between the elements and the intersection boundaries of objects.

The regular mesh obtained as a result of applying the algorithm makes it possible to perform a to perform finite element modeling of stress-strain state model of the human spine, the results of which can then be used to solve such problems as determining the risk and predicting the course of diseases of the musculoskeletal system, and selecting the optimal treatment strategy before surgery and evaluation of the effectiveness of measures taken in the treatment process.

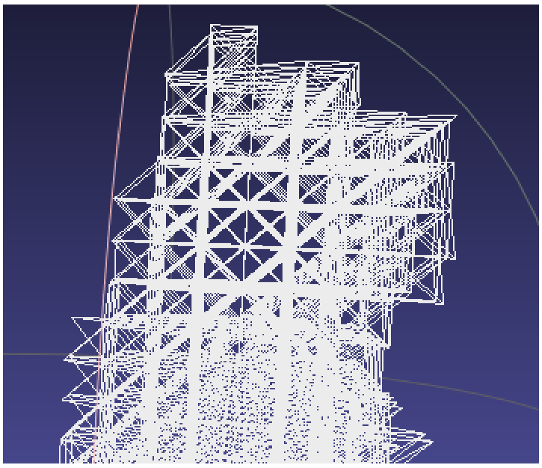

Figure 5. The results of the finite element mesh generation for a system of «vertebrae – intervertebral disc – vertebrae» based on stl-model.

IV. Verification of the Proposed Decomposition Algorithm

To verify the finite element mesh generated by the proposed algorithm and verify its suitability for solving real problems of stress-strain modeling, the simulation results in FreeFem software using the built-in irregular mesh generator were compared with the simulation results based on the mesh generated by the proposed algorithm.

As the first test case was considered a cylinder with a radius of 11 mm. and a height of 90 mm from the same material. Applied vertical load is concentrated at the top center of the cylinder plane. Fig. 6 presents the obtained displacement values for both meshes.

It can be seen from the graph that the displacement values for the resulting mesh and for the mesh from the FreeFem program are almost identical, the error does not exceed 3%.

V. Finding the Stress-Strain State of the Lumbar Spine

The proposed algorithm was tested on a real model of the human lumbar spine, reconstructed on the basis of medical images. The model contained two groups of elements with different material characteristics - L1-L5 vertebrae and intervertebral disks.

The material of the vertebrae is bone, possessing the following physical characteristics:

- Young's modulus 3432 MPa
- Poisson's ratio of 0.3
- Density of the fabric is 2020 kg/m^3

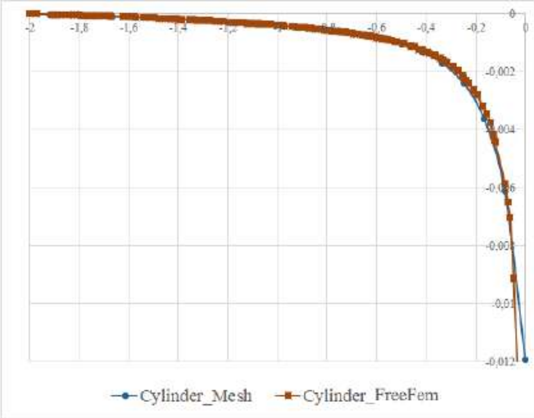

Figure 6. Comparison of the results of the calculation of the test model with the cylinder using its own mesh and mesh of FreeFem software

The material of the intervertebral disc has the following physical characteristics:

- Young's modulus 559 MPa
- Poisson's ratio of 0.4
- Density 1090.3 kg/m^3

A vertical uniformly distributed load is applied to the upper plane of the vertebra L1. Fig. 7 presents the obtained values of displacements in this model.

Figure 7. Graph of displacement in the lumbar spine model

From the graph one can see a clear dependence of the displacement values on the material of the object — a sharp change in displacements occurs at the border of intervertebral disks.

VI. Conclusion

Algorithm for constructing regular finite element mesh based on the three-dimensional stl-model «vertebrae - intervertebral disc - vertebrae» was developed. The algorithm is resistant to arbitrary irregularities of the surfaces of model objects, defects in objects and gaps between objects, which allows applying it to models of any quality.

The correctness of the algorithm is verified by comparing the simulation results based on the grid generated by the algorithm with the simulation results in the FreeFem software package.

References

[1] E.L. Tsitko, A.F. Smeyanovich, E.S. Astapovich, E.V. Tsitko, RRentgenometricheskii analiz kinematiki L4 - L5 i L5 - S1 pozvonochnykh segmentov v III stadii degenerativnogo protsessa, *Novosti Khirurgii*, 2015, vol. 23, pp. 202–208.

[2] C. Xiong, A. Suzuki, M.D. Daubs, T. Scott, K. Phan, J.Wang, The evaluation of cervical spine mobility without significant spondylosis by kMRI, *European Spine Journal*, 2015, vol. 24(12), pp. 2799-2806.

[3] E.L. Tsitko, K.S. Kurochka, N.N. Masalitina, I.N. Tsalko, V.V. Komrakov, E.V. Tsitko, Rentgenometricheskaya otsenka kinematiki poyasnichno-kresttsovogo otdela pozvonochnika pri osteokhondroze s pomoshch'yu programmnogo sredstva «Volot», *Problemy zdorov'ya i ekologii*, 2017, vol. 4 (54), pp. 35–41.

[4] K. S. Kurochka, T. V. Luchsheva, K. A. Panarin, Lokalizatsiya pozvonkov cheloveka na rentgenovskikh izobrazheniyakh s ispol'zovaniem Darknet YOLO, Minsk, *Doklady BGUIR*, 2018, vol. 3 (113). pp. 32–38.

[5] J. N. Reddy, *An Introduction to the Finite Element Method*, New York, McGraw-Hill Education, 2006, p. 784.

[6] Zhangxin Chen, *The Finite Element Method: Its Fundamentals and Applications in Engineering*, Singapore, World Scientific Publishing, 2011, p. 348.

[7] J. Schöberl, NETGEN An advancing front 2D/3D-mesh generator based on abstract rules, *Computing and visualization in science*, 1997, vol. 1, pp. 41–52.

[8] H. Si, TetGen. Tetrahedral mesh generator and threedimensional delaunay triangulator, *ACM Transactions on Mathematical Software*, 2015, vol. 41 (2), pp. 1–36.

[9] C. Geuzaine, J.F. Remacle, Gmsh: A 3-D finite element mesh generator with built-in pre- and post-processing facilities, *International Journal for Numerical Methods in Engineering*, 2009, vol. 79 (11), pp. 1309–1331.

[10] B.A. Christiansen, D.L. Kopperdahl, D.P. Kiel, T.M. Keaveny, M. L. Bouxsein, Mechanical contributions of the cortical and trabecular compartments contribute to differences in age-related changes in vertebral body strength in men and women assessed by QCT-based finite element analysis, *Journal of Bone and Mineral Research*, 2011, vop. 26(5), pp. 974–983.

АЛГОРИТМ ГЕНЕРАЦИИ КОНЕЧНОЭЛЕМЕНТНОЙ СЕТКИ ДЛЯ СИСТЕМЫ «ПОЗВОНОК – МЕЖПОЗВОНОЧНЫЙ ДИСК – ПОЗВОНОК» НА ОСНОВЕ STL МОДЕЛИ

Курочка К. С., Панарин К. А., Карабчикова Е. А.

Авторами предлагается устойчивый к дефектам модели алгоритм генерации регулярной конечноэлементной сетки на основе трехмерной stl-модели «позвонок – межпозвоночный диск – позвонок», позволяющий производить корректное разбиение модели при наличии пересекающихся объектов или объектов с не полностью соприкосающимися поверхностями. Получаемые конечные элементы различных объектов будут гарантированно совпадать, т.е стороны и ребра соседних элементов будут гарантированно повторять друг друга.

Received 30.12.18