Scientific Visualization, 2024, volume 16, number 1, pages 112 - 123, DOI: 10.26583/sv.16.1.10

Mathematical Modeling and Visualization of a Complex Stress State in Case of a Fracture of the Femoral Diaphysis

K.N. Krupin[1,A], M.A. Kislov[2,B], V.I. Bahmetev[3,C], E.M. Kildyushov[4,A]

[A] Pirogov Russian Research Medical University, Moscow, Russia
[B] Sechenov First Moscow State Medical University (Sechenov University), Moscow, Russia
[C] Voronezh State Medical University named after N.N. Burdenko, Voronezh, Russia

[1] ORCID: 0000-0001-6999-8524, krupin_kn@rsmu.ru
[2] ORCID: 0000-0002-9303-7640, smedik@gmail.com
[3] ORCID: 0000-0002-8770-1664, bahmetev@vsmaburdenko.ru
[4] ORCID: 0000-0001-7571-0312, kem1967@bk.ru

Abstract

The purpose of this work is to establish the possibility of using the finite element analysis method to study complex stress states in case of a femur fracture with subsequent data visualization. Experimental data were obtained on a solid-state mathematical parametric model of the femur, created on the basis of computer tomogram data, and repeating studies on native biological objects. As a result of mathematical modeling, oblique transverse and helical fractures of the diaphysis of the femur with elements of helical deformation were studied. The application of finite element analysis made it possible to visualize and predict the stresses arising in bone tissue under the impact of a blunt solid object in a complex stress state and the morphological features of femoral shaft fractures under different torsional loading forces of the proximal part of the femur. The data on the mechanism and morphology of the femoral shaft fracture obtained during modeling are confirmed by the results of original full-scale experiments.

Keywords: femur, impact, complex stress state, finite element analysis, forensic medicine.

1. Introduction

The high prevalence of musculoskeletal injuries, reaching 24% in the structure of temporary disability, and the high incidence of femoral fractures indicate the relevance of studying the mechanism of their formation [1]. However, most scientific works describe the mechanism of fracture formation in one type of deformation. The criteria for establishing the mechanism of damage formation obtained in such works are used in forensic medical examinations using the method of loose analogy. In real life, such simple conditions of loading the femur are rare, therefore the morphology of a real fracture has multiple additional signs that are not typical for one type of deformation, which raises reasonable doubts among the expert about his correctness in establishing the mechanism of injury and requires the development of new methods for establishing the mechanism of injury.

Our previous experiments on mathematical modeling using the finite element analysis method, which is widely used to solve problems in the mechanics of a deformable solid in the science of “strength of materials”, of femur fractures allowed us to validate a solid parametric model of the femur created from computed tomography data of real people and apply it to study the mechanism of formation of femoral fractures in various conditions and circumstances [2].

The purpose of this work is to establish the possibility of using the finite element analysis method to study complex stress states during a femur fracture with subsequent visualization of the results.

2. Research methods

During the preliminary part of the study, literature sources were analyzed that examined the morphology of fractures of the diaphysis of long tubular bones caused by the impact of blunt objects. This analysis was based on data obtained from experiments on biomannequins or during practical observations (342 experiments on 200 biomannequins, 56 experiments on lower limb bone samples, 116 expert observations) [3]. The femoral loading conditions and failure modes used in the model were taken from these studies to verify the validity of the model. In the experiments, impact on biomannequins was carried out with a horizontal position of the body on a metal surface and a blow with a blunt hard object on the anterior surface of the diaphysis of a long tubular bone, while the distal part of the femur was fixed bone and pressure was applied to the pelvis to simulate the load of rotation of the upper femur clockwise or counterclockwise. The impact force causing fractures in the experiments ranged from 1800 N to 1900 N [3].

To test the possibility of mathematical modeling of the process of formation of a fracture of the femur, the finite element method (FEM) was used, implemented using the ANSYS LS-DYNA software environment, which is a popular program for finite element analysis, developed by Ansys inc., which is used to solve linear problems. and nonlinear dynamic problems of mechanics of deformable solids, including fracture analysis (ANSYS inc., https://www.ansys.com) [4]. In this work, ANSYS LS-DYNA was used to reproduce experiments conducted on biomannequins and during practical observations under impact impacts. To recreate the loading conditions of the hip model, fastening was performed in the area of the articular surfaces with a rigid base and the formation of an elastic substrate simulating the Winkler base. The finite element mesh was automatically generated using 5 mm Solid elements to simulate the volumetric stress-strain state.

The movement of the bone along the Z axis is limited in the proximal (upper) part of the bone (hip joint area) and in the distal (lower) part of the bone (knee joint area). The preload was modeled by applying a force of 100 N to the surface of the femoral head and directed in the first experiment along the Y axis, and in the second – against the Y axis. The force was applied at an angle of 90 degrees to the axis of the bone of the femur model in the projection of the middle third of the anterior surface of the femur and was implemented a simulated cylindrical steel indenter with a radius of curvature of 30 mm and a length of 180 mm. Moreover, its movements are limited in all directions except vertical. The impact was modeled at an impact speed of 18 m per second. The following influence conditions are specified between the parts of the model:

- between spongy substance, compact substance and muscle tissue – conditions of inextricable connection, which is achieved by constructing a conformal mesh of finite elements on these parts;
- between the muscle tissue, the punch and the base – contact with friction.

Considering that when conducting experiments on biomannequins and in practical observations, the properties of bone and soft tissues were not studied, averaged mechanical properties of tissues from various literature sources (see Tables 1 and 2) [5-8]. Such tissue characteristics described well the behavior of materials in previous experiments [2]. The properties of bone tissue components are described by material models – isotropic elastoplastic for spongy substance and anisotropic elastic for compact substance. The theory of maximum principal stress (Tresca theory) was adopted as a destruction model.

***Table 1*. Mechanical properties of a compact bone tissue**

Density	2,000 kg/m^3
Young’s modulus X direction	12 GPa
Young’s modulus Y direction	12 GPa
Young’s modulus Z direction	20 GPa
Poisson’s ratio XY	0.38
Poisson’s ratio YZ	0.22
Poisson’s ratio XZ	0.22
Shear modulus XY	4.5 GPa
Shear modulus YZ	5.6 GPa
Shear modulus XZ	5.6 GPa
Compressive ultimate strength	0.205 GPa
Tensile ultimate strength	0.133 GPa
Maximum tensile stress	52 MPa
Maximum shear stress	65 MPa

***Table 2*. Mechanical properties of sponge bone tissue**

Density	127 kg/m^3
Young's modulus	0.38 MPa
Poisson’s ratio	0.33
Bulk modulus	0.37255 MPa
Shear modulus	0.14286 MPa
Compressive ultimate strength	6.23 MPa
Tensile ultimate strength	8.4 MPa
Maximum tensile stress	8.4 MPa
Maximum shear stress	7.4 MPa
Yield strength	1.75 MPa
Tangent modulus	41.8 MPa

Due to the fact that in our model, the destruction of soft tissues was not studied, and the physical properties of soft tissues were generally used only as a spacer between the indenter and the base, the mechanical properties of soft tissues were taken from the properties of the ballistic gel. The Johnson-Cook model was adopted as a fracture model, taking into account changes in strength criteria depending on loading rate and temperature (see Table 3):

***Table 3*. Mechanical properties of soft tissues**

Density	1.030 kg/m^3
Young’s modulus	59.862 kPa
Poisson’s ratio	0.4956
Bulk modulus	29 kPa
Shear modulus	20 kPa
Johnson Cook failure	
Damage constant D1	−0.13549
Damage constant D2	0.6015
Damage constant D3	0.25892
Damage constant D4	0.030127
Damage constant D5	0
Melting temperature	20°C
Reference strain rate (/sec)	1

3. Results and discussion

Visualization of the research results was obtained in the ANSYS LS-DYNA postprocessor.

In experiment 1, when preloading in the proximal part of the femur with a value of 100 N along the Y axis, as a result of modeling an impact perpendicular to the longitudinal axis of the femur, an oblique transverse asymmetrical fracture line was formed in the conditional middle of the femoral diaphysis with chipping of elements in the fracture line and the beginning of the formation of a fracture line in the area of minimum thickness of the compact bone substance in the injured area proximal to the site of application of the traumatic force of the femur on the opposite side of the load. Maximum equivalent voltages reached 52.958 MPa (Fig. 1-3). When analyzing the principal stress vectors (Fig. 3), a dynamic distribution of compressive (blue arrows) and tensile (red arrows) stresses is noted, changing their location and magnitude depending on the stage of bone destruction. At the initial stage of loading, asymmetric stresses arose in the proximal part of the bone, indicating a helical deformation of the bone, not exceeding the limits of structural destruction that occur during preloading (Fig. 4).

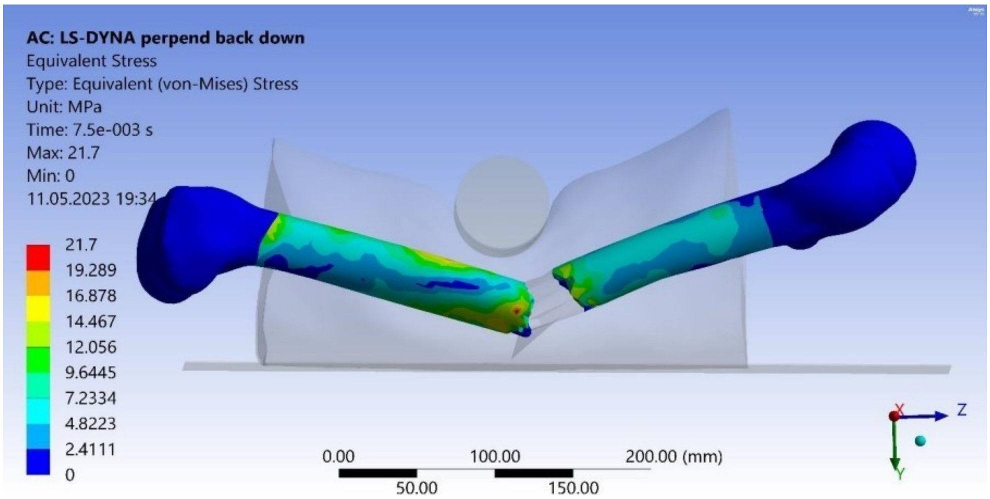

Fig. 1. Visualization of the final stage of modeling a fracture of the femoral diaphysis in a complex stress state with preload in the proximal part of the femur along the Y axis. Von Mises analysis.

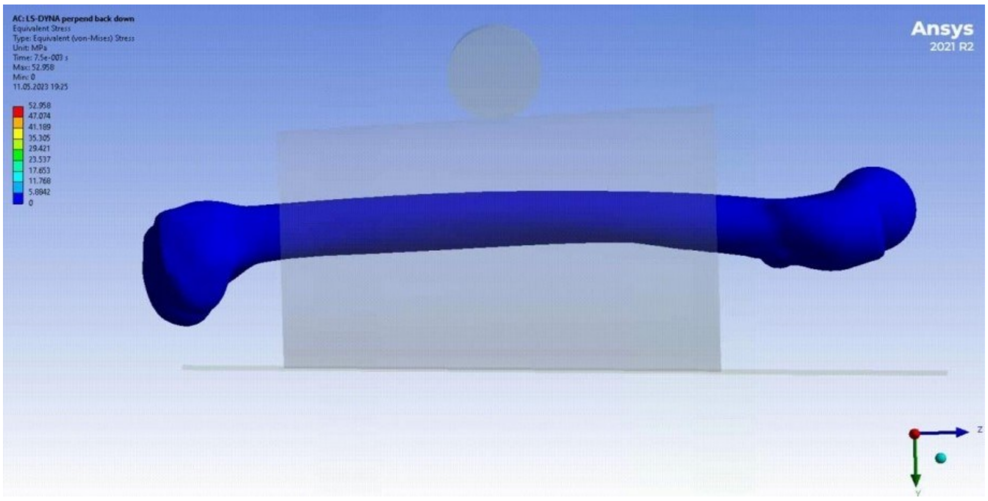

Fig. 2. Dynamic visualization of modeling a fracture of the femoral diaphysis in a complex stress state with preload in the proximal part of the femur along the Y axis. Von Mises analysis.

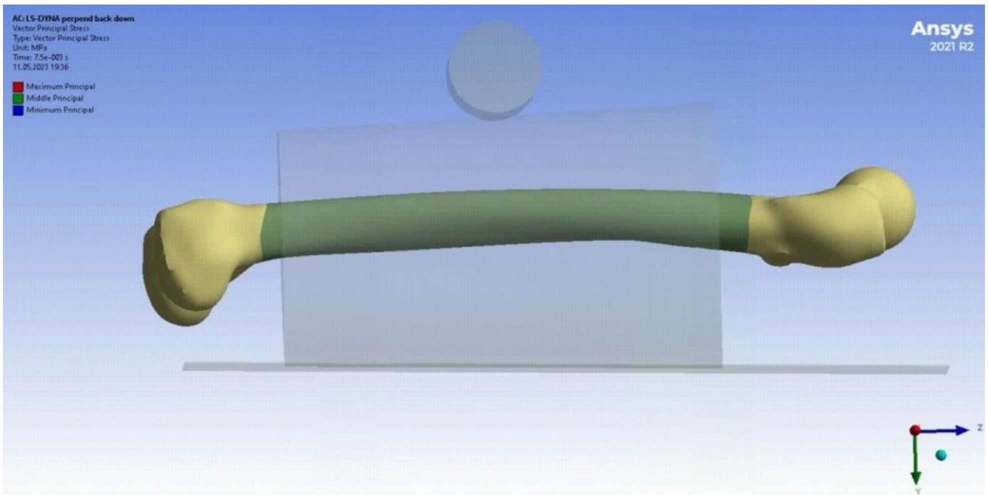

Fig. 3. Dynamic visualization of modeling a fracture of the femoral diaphysis in a complex stress state with preload in the proximal part of the femur along the Y axis. Analysis using principal stress vectors.

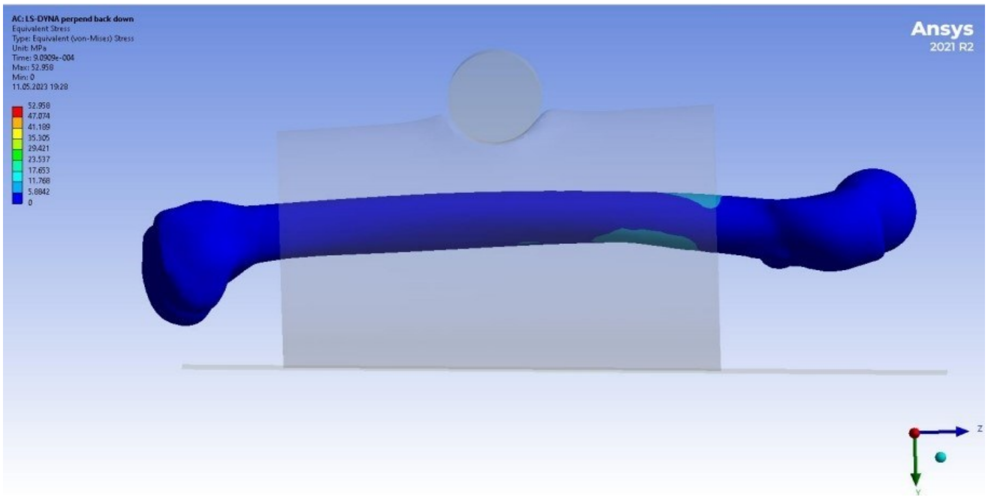

Fig. 4. Visualization of internal stresses in the proximal part during torsional deformation of the upper part of the femur with preload in the proximal part of the femur along the Y axis.

In experiment 2, with a preload of 100 N in the proximal part of the femur directed in the opposite direction of the Y axis, as a result of modeling the impact perpendicular to the longitudinal axis of the femur, an oblique transverse asymmetrical fracture line was formed in the conditional middle of the femoral diaphysis with chipping of elements in the fracture line and the beginning of formation the fracture line in the area of minimum thickness of the compact bone substance in the injured area proximal to the site of application of the traumatic force of the femur on the opposite side of the load. The maximum equivalent stresses reached 59.218 MPa (Fig. 5-7). When analyzing the principal stress vectors (Fig. 7), a dynamic distribution of compressive (blue arrows) and tensile (red arrows) stresses is noted, changing their location and magnitude depending on the stage of bone destruction. At the initial stage of loading, asymmetric stresses arose in the proximal part of the bone, indicating a helical deformation of the bone, not exceeding the limits of structural destruction that occur during preloading (Fig. 8).

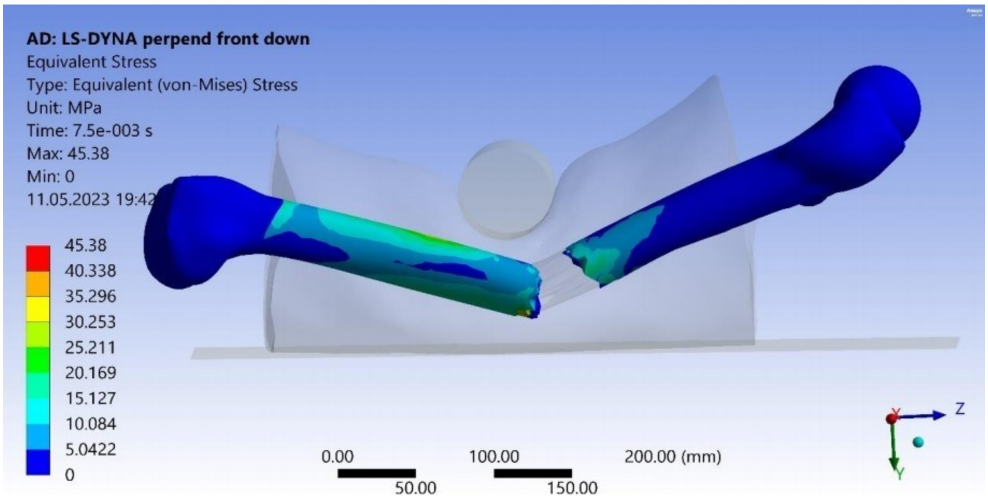

Fig. 5. Visualization of the final stage of modeling a fracture of the femoral diaphysis in a complex stress state with preload in the proximal part of the femur in the opposite direction of the Y axis. Von Mises analysis.

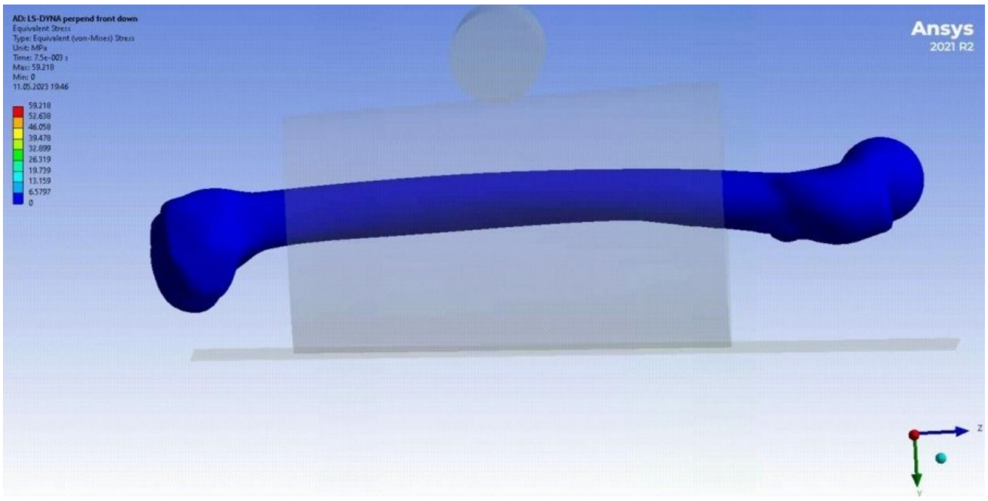

Fig. 6. Dynamic visualization of modeling a fracture of the femoral diaphysis in a complex stress state with preload in the proximal part of the femur in the opposite direction of the Y axis. Von Mises analysis

.

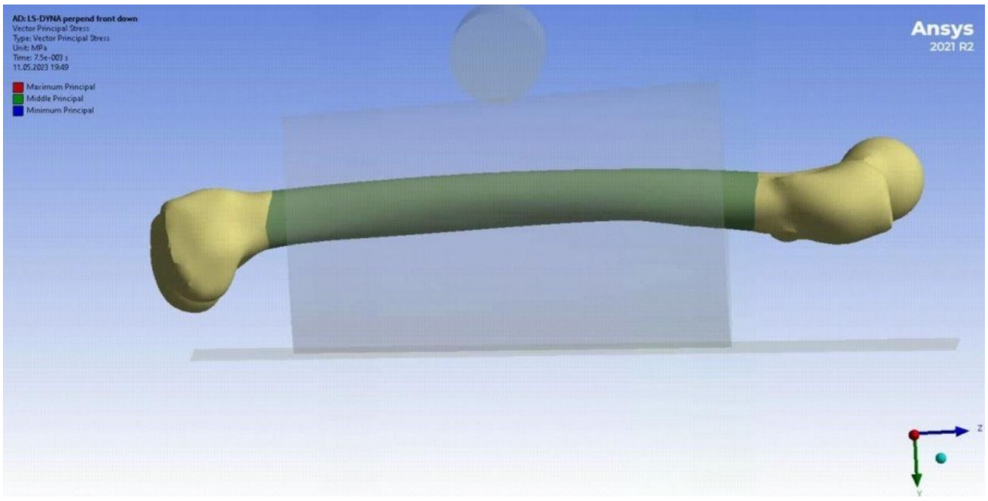

Fig. 7. Dynamic visualization of modeling a fracture of the femoral diaphysis in a complex stress state with preload in the proximal part of the femur in the opposite direction of the Y axis. Analysis by principal stress vectors.

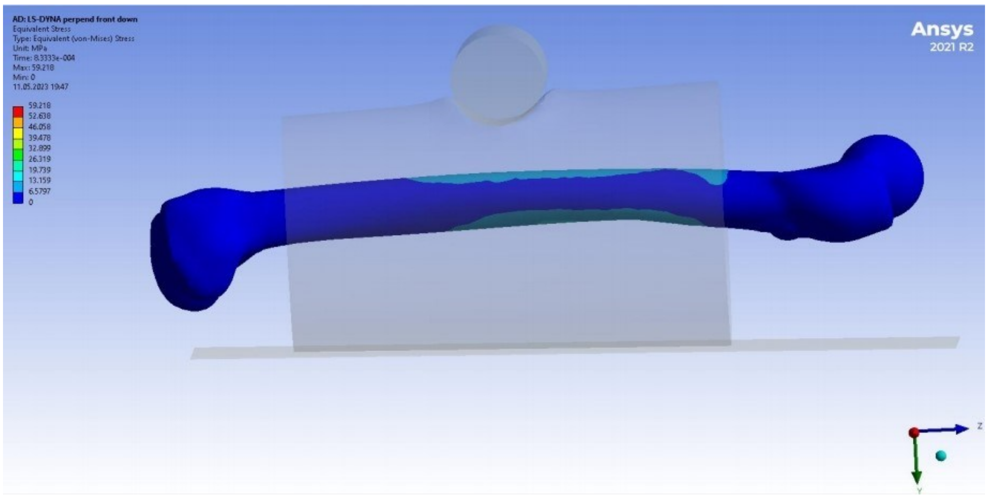

Fig. 8. Visualization of internal stresses in the proximal part during torsional deformation of the upper part of the femur with preload in the proximal part of the femur in the opposite direction of the Y axis.

To validate the mathematical model, we compared the morphologies of fractures that occur under similar conditions on biomannequins and in practical observations, in which the fracture line arose in the area of action of a traumatic object with the formation of an oblique-transverse asymmetric fracture line with typical signs of bone tissue stretching on the opposite side from the site of impact. In some observations, an additional helical fracture line was formed on the proximal part of the bone, which is closer to the area of bone rotation [3, 10] (Fig. 9-11).

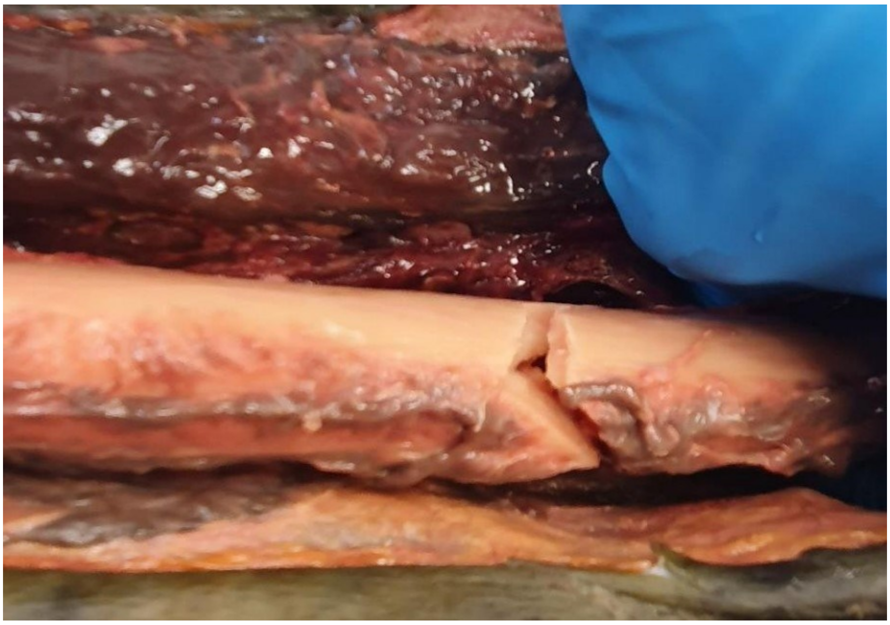

Fig. 9. Image of the fracture line of the femoral diaphysis at the site of local impact trauma.

Fig. 10. Image of the fracture surface of the femoral diaphysis at the site of local impact trauma with pronounced asymmetry.

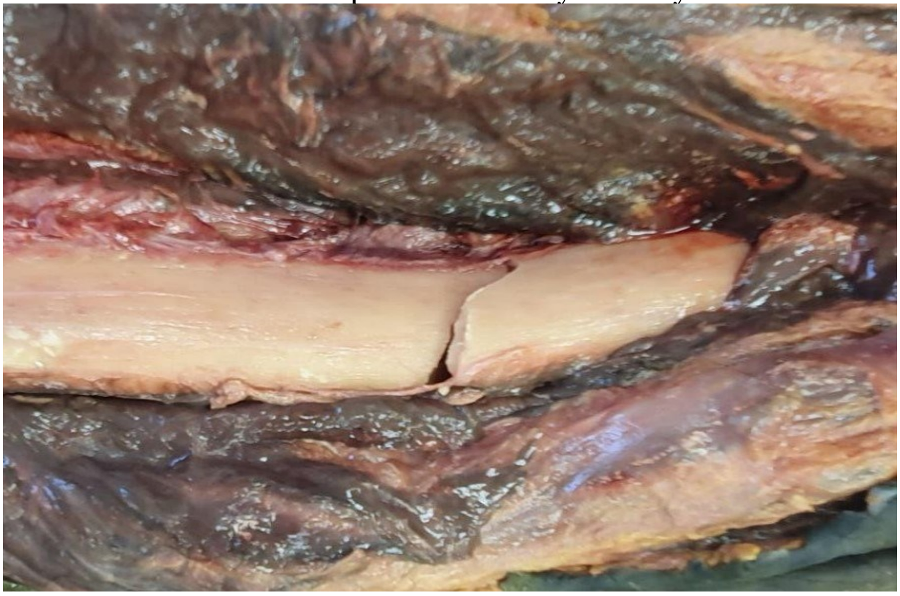

Fig. 11. Image of a helical fracture line of the proximal part of the femoral diaphysis outside the site of local impact trauma.

In the described natural experiments, the properties of bone tissue were not studied; only the absence of any pathology of bone tissue was noted. Also, in experiments on biomannequins and practical observations, the change in the thickness of the compact bone plate along the diaphysis of the femur in the area of traumatic influence was not studied.

When analyzing experimental data and practical observations, we assumed that the formation of a helical fracture line in the proximal part of the femur occurs when the stresses of the bone tissue exceed its strength characteristics. Therefore, it was decided to conduct an additional experiment by applying more force to the proximal femur under preload and removing the rigid support without changing the remaining boundary conditions.

In experiment 3, with a preload in the proximal part of the femur of 300 N along the Y axis as a result of modeling the impact perpendicular to the longitudinal axis of the femur in the conditional middle of the diaphysis of the femur, an oblique-transverse asymmetrical fracture line has formed with chipping of elements in the fracture line and the beginning of the formation of a fracture line in the area of minimum thickness of the compact bone substance in the injured area proximal to the place of application of the traumatic force of the femur on the opposite side from the loading and a helical fracture line in the proximal part of the femoral diaphysis . The maximum equivalent stresses reached 119.95 MPa (Fig. 12-14).

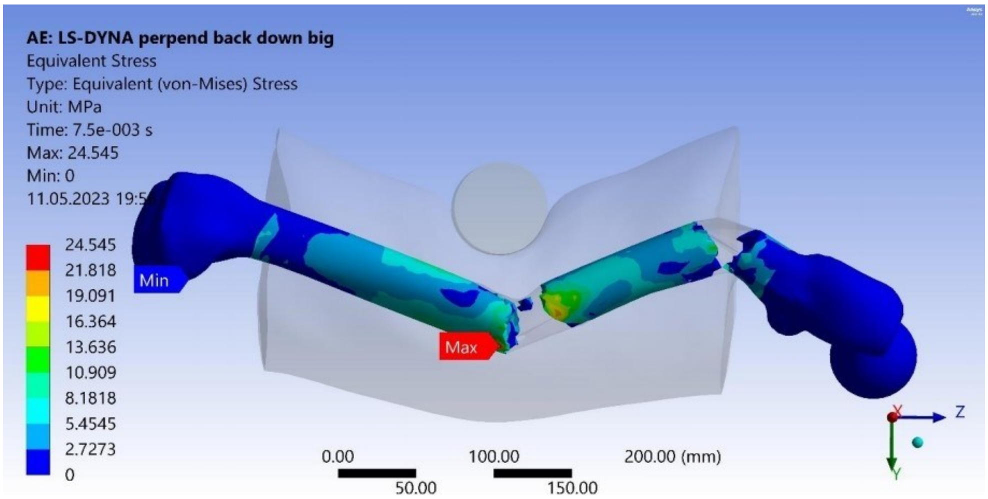

Fig. 12. Visualization of the final stage of modeling a fracture of the femoral diaphysis in a complex stress state with a preload increased to 300 N in the proximal part of the femur along the Y axis. Von Mises analysis.

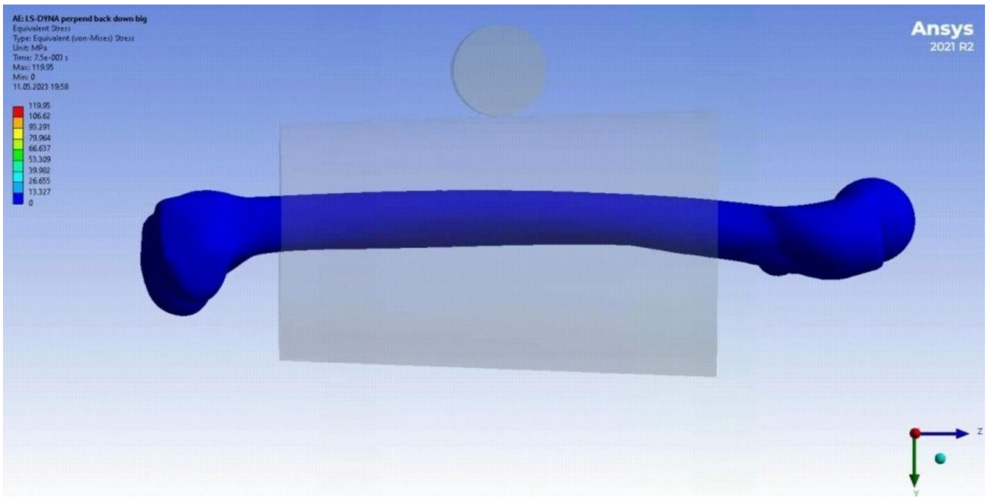

Fig. 13. Dynamic visualization of modeling a fracture of the femoral diaphysis in a complex stress state with a preload increased to 300 N in the proximal part of the femur along the Y axis. Von Mises analysis.

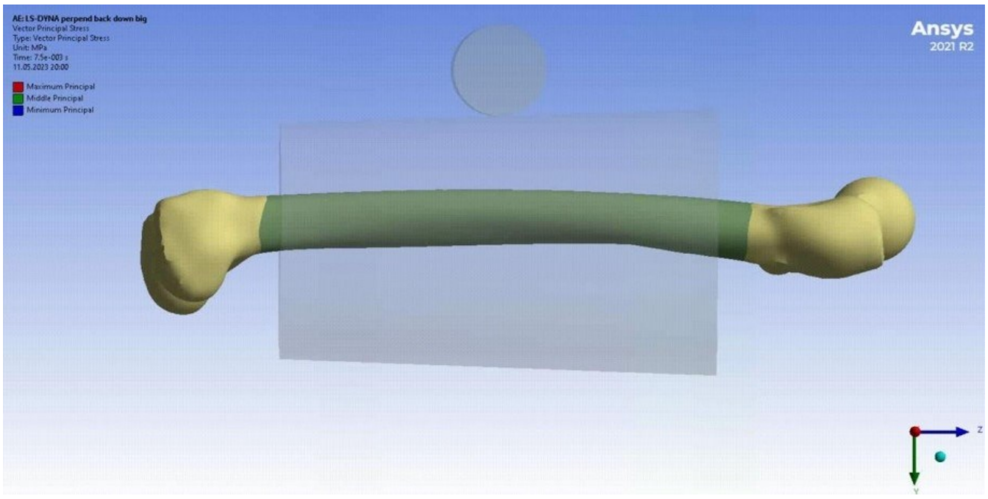

Fig. 14. Dynamic visualization of modeling a fracture of the femoral diaphysis in a complex stress state with preload increased to 300 N in the proximal part of the femur along the Y axis. Analysis using principal stress vectors.

When comparing the morphological features of fractures and their modeling in our experimental study, we found a correspondence between the localization and nature of fractures. However, the variability of the morphological features of femoral shaft fractures in our experimental study using mathematical modeling had a pattern depending on:

- direction of the action vector of the traumatic force and the magnitude of the preload,
- places of application of traumatic force (local fracture occurs in the area application of a traumatic force with a rupture area on the opposite side, a helical fracture occurs with significant rotational deformation closer to the rotation area),
- thickness of the compact plate in the area of traumatic impact on the opposite side from the site of application of the traumatic force.

4. Conclusion

Finite element analysis allows us to visualize and predict the stresses that arise in bone tissue under a combination of impact and rotation (complex stress state). The data obtained during modeling are confirmed by the results of original field experiments and practical observations.

Using mathematical modeling, the dependence of the morphological characteristics of fractures on the place of application of the traumatic force on the anterior surface of the thigh, the thickness of the compact plate, and the place of application of rotating loads was revealed. When a combination of impact and rotation effects occurs in a local fracture, a combination of signs of angular deformation with asymmetry of the fracture plane, characteristic of a helical fracture, appears. As rotation of the femur increases, an additional helical fracture line appears closer to the point of rotation.

The experiments performed show the possibility of using the finite element method in forensic medicine, which makes it possible to reliably predict the process of destruction of biological objects under various types of mechanical impact and visualize its results at any stage of the experiment with a large amount of displayed data. In the future, it is possible to use this information to solve the inverse problem – determining the trace evidence properties of a traumatic weapon based on the morphological picture of destruction. This confirms the high efficiency of finite element analysis in forensic medicine.

References

1. Streubel PN, Gardner MJ, Ricci WM. Management of femur shaft fractures in obese patients. *Orthopedic Clinics of North America*. 2011 Jan;42(1):21-35. https://doi.org/10.1016/j.ocl.2010.07.004
2. Kislov M.A., Bakhmetyev V.I., Kildyushov E.M., Krupin K.N. Matematicheskoye modelirovaniye pereloma diafiza bedrennoy kosti pod ostrym uglom. Sudebno-meditsinskaja ekspertiza. 2022;65(6):37-41. [In Russian]. http://doi.org/10.17116/sudmed20226506137.
3. Bakhmetyev V.I., Kislov M.A., Stepanyan N.A. Osobennosti razrushenij kostnoj tkani pri razlichnyh sposobah vneshnego vozdejstvija / Sistemnyj analiz i upravlenie v biomedicinskih sistemah. 2006. No. 5(1). P. 192-195 [In Russian].
4. Oficial'naja stranica produkta Ansys LS-DYNA. Data obrashhenija 15.05.22. https://www.ansys.com/products/structures/ansys-ls-dyna
5. Kopperdahl DL, Keaveny TM. Yield strain behavior of trabecular bone. *J Biomech*. 1998;31(7):601-608. https://doi.org/10.1016/s0021-9290(98)00057-8
6. Yosibash Z, Tal D, Trabelsi N. Predicting the yield of the proximal femur using high-order finite-element analysis with inhomogeneous orthotropic material properties. Philosophical Transaction of the Royal Society a Mathematical, Physical and Engineering Sciences. 2010;368(1920):2707-2723. https://doi.org/10.1098/rsta.2010.0074
7. Keller TS, Mao Z, Spengler DM. Young's modulus, bending strength, and tissue physical properties of human compact bone. Journal of Orthopaedic Research. 1990;8(4):592-603. https://doi.org/10.1002/jor.1100080416
8. Chirchir H. Limited Trabecular Bone Density Heterogeneity in the Human Skeleton. Anatomy Research International, 2016, 1(1):7. https://doi.org/10.1155/2016/9295383
9. Bahmetyev V.I. Opredelenie vida i napravlenija vneshnih vozdejstvij po morfologii razrushenija dlinnoj trubchatoj kosti. *Sudebnaja medicina*. 2019;5(S1):155. [In Russian].
10. Bakhmetyev V.I., Kislov M.A. Opredeleniye vida vneshnego vozdeystviya na osnove analiza morfologii izloma dlinnykh trubchatykh kostey nizhnikh konechnostey. Sudebno-meditsinskaya ekspertiza. 2008;51(6):11-14. [In Russian].