Fine Modeling and Mechanical Analysis of Human Lumbar Spine

Hongwei Zhang
School of Mechanics and Architecture Engineering, Jinan University, Guangzhou, Guangdong, China
DOI: 10.32629/jcmr.v5i1.1793

Abstract: This paper has created a skeletal model of the human lumbar spine and proved its effectiveness. Simulated scenarios when the human body is moving, including forward bending, backward extension, left bending, and left rotation. Compare range of motion, vertebral displacement, annulus fibrosus displacement, endplate displacement, nucleus pulposus displacement, annulus fibrosus stress, endplate stress, nucleus pulposus stress, and cortical bone stress. The model of this study was based on anatomical principles for detailed drawing of the human lumbar spine. ROMs under different physiological motions including flexion, extension, and lateral bending with 300N preload and 3.75N·m moment were measured under the normal finite element model. The degrees of flexion of L1-S1 were 17.204°. The degrees of extension of L1-S1 were 13.959°. The degrees of lateral bending of L1-S1 were 10.326°, axial rotation were 6. 466°. The maximum stress for intervertebral disc flexion is 1.4285MPa. The maximum stress of the extension intervertebral disc is 1.1296MPa. The maximum stress of the intervertebral disc with lateral bending is 1.7589MPa. The maximum stress of the axial rotating intervertebral disc is 1. 1698MPa. After comparing with classical literature, the model of this study meets clinical research standards and may be a good choice for clinical surgical analysis.
***Keyword*:** finite element model, stress cloud map, lumbar spine structure, intervertebral disc, ANSYS, biomechanics

1. Introduction

The human spine is an important structure that makes up the human body. In previous medical fields, doctors could only deepen their understanding of the internal structure of the human body through anatomy. When performing surgery on different patients, they could only have a rough understanding of the patient's spine. This has led to a lack of specificity in clinical surgery, especially for patients with special growth and development, or those with special disease locations, Most doctors conduct temporary analysis through clinical surgery without specific knowledge, which can lead to increased probability of surgical failure and postoperative complications.

The finite element method is a technique based on finite elements, which was originally applied to the analysis of material prestress in civil construction and aviation engineering, as well as computer simulation methods for analyzing metal material fatigue. [1, 2], Nowadays, with the changes of times and the continuous progress of technology, finite element method has been applied in various fields of life, and finite element analysis in medicine is booming. Some use finite element analysis to analyze the stress of different types of bones, so that doctors can design surgical procedures and different surgical methods. Some use finite element analysis to analyze the stress of different medical auxiliary devices in surgery, so as to facilitate preoperative, intraoperative, and postoperative surgical analysis and help prevent postoperative complications. The above finite element applications in the medical field only introduce a part of them, and there are still more application methods waiting for us to explore and research[3-6].

In clinical surgery, many patients with lumbar spine diseases are encountered, and different lumbar spine diseases are accompanied by different symptoms. Currently, the mainstream surgical methods in clinical surgery mostly involve removing the lesion and implanting different types of artificial implants. The effects of different implants vary, but before implanting them into the patient's body, they can only simulate the state outside the patient's body through experiments, and cannot accurately predict the situation of the implant in the patient's body, Therefore, the mechanical finite element analysis of surgical medical devices in clinical medicine has played a huge role. Mechanical finite element analysis can analyze the stress distribution trend of implants during clinical surgery, help verify the impact of implants on patients, verify the rationality of implants, prevent the impact of implants on postoperative life of patients, and prevent incorrect placement of implants from causing secondary harm to patients[7].

In recent years, with the continuous development of chip technology, the level of intelligence in computer products has steadily improved, and finite element technology has also made rapid progress. In production and life, people's requirements for finite element technology are constantly increasing, and the requirements for finite element accuracy are also becoming increasingly strict. The accuracy of finite element analysis depends on the size of the mesh division. A larger grid indicates a decrease in accuracy, a smaller grid indicates an improvement in accuracy, but with it comes an extension of computation time and an increase in computer performance requirements. [8]. Fortunately, in more conventional finite element analysis, engineers can make subtle adjustments to the size of the finite element mesh for different details in different models, which is also one of the ideas used in the model building process of this article.

Prior to this, many predecessors had already done a lot of related work on the biomechanical finite element of the lumbar spine, including different experimental methods in vivo and in vitro, which also provided a lot of reference for this paper. In 1994, Shirazi Adl [9] developed a finite element model of the first to fifth segments of the lumbar spine, which specifically considered axial compression, buckling, as well as lateral bending tensile and axial torsional moments. In his article, this finite element model maintains a certain degree of accuracy in finite element simulation of lumbar multi segment motion systems. Kiapour, Ambati et al. [10] created an L3-S1 lumbar spine FE model. They validated it based on a male CT sample and experimental data from literature[10-12]. Lin et al. [13] established an L1-L5 lumbar spine FE model and used it to study the effect of dynamic spinal fixators on the lumbar spine. Their model used CT samples from 19-year-old healthy male subjects and was compared and validated through experimental data in the literature[9, 14-17]. This finite element experiment is an intersection of mechanics and orthopedic medicine. The use of mechanical finite element analysis to analyze surgical cases under different working conditions has gradually become a new trend in the current medical industry. Mechanical finite element analysis can effectively guide clinical surgery and postoperative effects[18].

Traditional biomechanical research requires the use of animal or human cadaver samples for learning and studying human body parts, and it is also difficult to understand the stress, strain, and displacement of various details in the lumbar spine through cadaver models. Even if the corresponding data is measured, it may be biased due to laboratory environmental factors; However, the model established by traditional finite element analysis is relatively rough and lacks details. The model of this study collaborated with different software to establish a detailed model of the human lumbar spine. The ligaments used in the model were linear ligaments that were only subjected to tension. There are detailed distinctions and studies on the endplate and fibrous ring nucleus pulposus in intervertebral discs, including the detailed establishment of the sacrum and the differentiation of cortical and trabecular bone. The number of ligaments selected for establishment varies in different locations.

In medicine, knowledge in the field of mechanics is usually not accessible, and mechanical knowledge can play a crucial role in medicine. Mechanical analysis can consider mechanical issues that doctors cannot consider during surgery. For example, mechanical finite element analysis can analyze the rationality of surgical plans based on the shape of the spine, the material of internal fixation, and the position of internal fixation. Changes in the boundary conditions of the bone after implant surgery can cause stress concentration, and if such issues are not noticed after surgery, it may cause secondary harm to the patient. The original intention of establishing this model is to explore the optimal surgical methods for different patients using finite element analysis and provide them with the best treatment plan. This article will introduce the finite element modeling of the human lumbar spine and conduct relevant biomechanical analysis. (Figure 1)

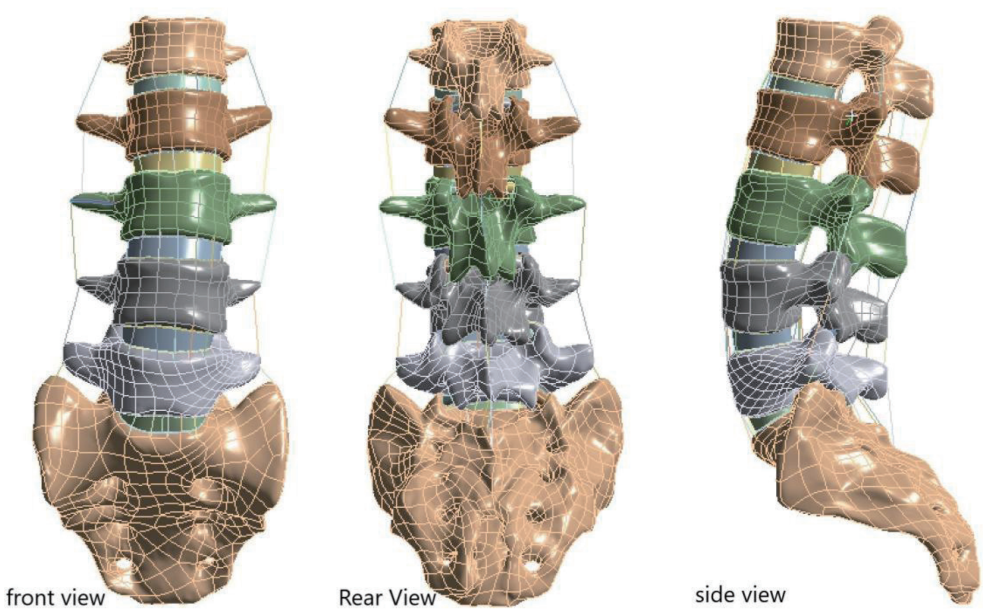

Figure 1. Three views of the lumbar spine

2. Materials and methods

2.1 Data Acquisition

The study was approved by the author's affiliated institutions ethics committee. Lumbar data were derived from a 23-year-old healthy male volunteer who had no spinal-related diseases through clinical and radio graphic examination.

2.2 Construction of Lumbar Finite Element Model

The images were segmented using Mimics to obtain the boundaries of the skeleton. Due to the presence of uneven surfaces, holes that cannot be autonomously corrected by software, and sharp protrusions in the extracted model, so Geomagic Studio (2021) (Geomagic, Inc, Research Triangle Park, NC, United States) is used to polish. Polish the bone model extracted from Mimics to make it smooth and even for subsequent mesh partitioning and finite element analysis. Import the polished vertebral body parts into Solidworks 2021(SolidWorks Corporation, MA, United States). The annulus fibrosis, nucleus pulposus, endplate, and articular cartilage were established in Solidworks 2021 according to the anatomical characteristics of the lumbar spine. In this study, the thickness of cortical bone is 2mm and the thickness of endplate is 0.5mm[19].

2.3 Finite element model of L1-S1

In this study, the material properties of cortical cancellous bone joint surfaces and endplates were defined as isotropic, uniform, and linear elastic materials. All contact surfaces were considered bounded except for the small joint contact surface set to be frictionless. The seven main ligaments are anterior longitudinal ligament (ALL), posterior longitudinal ligament (PLL), ligamentum flavum (LF), capsular ligament (CL), intertransverse ligament (ITL), interspinous ligament (ISL), and supraspinous ligament (SSL), with ligament units set as rod units that are only subjected to tension. The finite element model used in this study includes lumbar cortical bone, cancellous bone, intervertebral disc, endplate, and ligament. The intervertebral disc is divided into fibrous annulus, nucleus pulposus, upper and lower endplates. The ligaments in this model are established as linear ligaments, and corresponding Young's modulus and Poisson's ratio are assigned in the model. As shown in Table 1, Grid them and allocate material parameters. Material parameters for various organizations are taken from Zhang et al. [20].

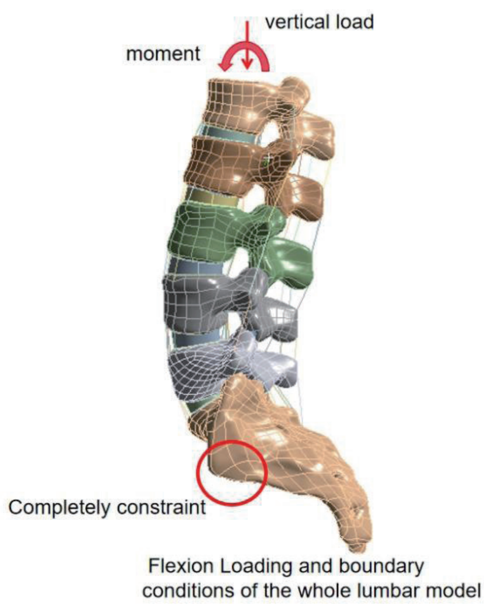

Figure 2. Lumbar spine boundary conditions

The loading process first applies a vertical load of 300N to simulate the weight of the upper body of a normal person in an upright position. Secondly, a torque of 3.75N·M[20] is applied to the L1 surface to test the 4 movement scenarios of the lumbar spine model. Including flexion, extension, lateral bending, and axial rotation. The degrees of freedom at the bottom of the S1 surface are constrained in all directions. (Figure 2)

2.4 Material parameters

After investigation, the material performance data selected for this finite element experiment are shown in Table 1 below, and the material parameters of each organization are taken from literature.

Table 1. Finite element model simulation of lumbar spine with various material property parameters

	Young's Modulus (MPa)	Poisson ratio	Cross-section area(mm²)
Cortical bone	12000	0.3	
Cancellous bone	100	0.3	
Endplate	25	0.25	
Annulus fibrosis	4.2	0.45	
Nucleus pulposus	1	0.499	
Anterior longitudinal	7.8	0.3	22.4
Posterior longitudinal ligament	10	0.3	7.0
Ligamentum flavum	15	0.3	14.1
Capsular ligament	7.8	0.3	10.5
interspinous ligament	8	0.3	0.6
Supraspinous ligament	8	0.3	10.5
Intertransverse ligament	10	0.3	14.1

3. Model Validation and Biomechanical Evaluation

A FE study on the lumbar spine requires several simplifications and assumptions with fundamental impact on the results. Therefore, experimental verification of the model is needed[21]. In a spine tester, Rohlmannet have loaded lumbar spines with pure moments of 3.75N·m[20] acting in the three main planes simulating flexion/extension, left and right lateral bending, and left and right axial rotation[22]. The L1 vertebra was fixed and the moments were applied at Ll vertebra. This study not only compared with in vivo experiments but also with [23]lumbar spine FE data to verify the validity of the model. With fixation on the inferior surface of the sacrum, a 300N preload was imposed on the superior surface of L1 and a 3.75N·m moment was applied on the L1 superior surface to simulate 4 different physiological motions: flexion, extension, bending, and torsion for model validation.

The range of motion (ROM) of the entire lumbar spine at Ll-S1 under 4 conditions of pure torque of 3.75NM was recorded and the results were compared with in vitro experimental data reported by Rohlmann[22] and FE simulation data by Zander[23]. In this study, the ROM obtained by the FE model is basically within the standard deviation of the average value of the in vitro experiments, as shown in Figure 3[20]. This model has the validity of finite element simulation experiments. (Figure 3)

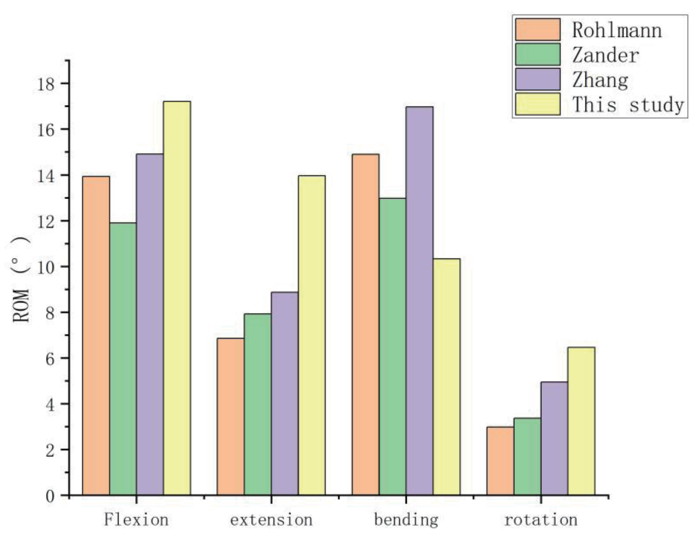

Figure 3. Comparison of data with study of Rohlmann, Zander and Zhang

4. Results

ROMs under different physiological motions including flexion, extension, and lateral bending with 300N preload and 3.75N·m moment were measured under the normal finite element model. The degrees of flexion of L1-S1 were 17.204°. The degrees of extension of L1-S1 were 13.959°. The degrees of lateral bending of L1-S1 were 10.326°, axial rotation were 6.466°.

The maximum stress for intervertebral disc flexion is 1.4285MPa. The maximum stress of the extension intervertebral disc is 1.1296MPa. The maximum stress of the intervertebral disc with lateral bending is 1.7589MPa, The maximum stress of the axial rotating intervertebral disc is 1.1698MPa. (Figure 4~10)

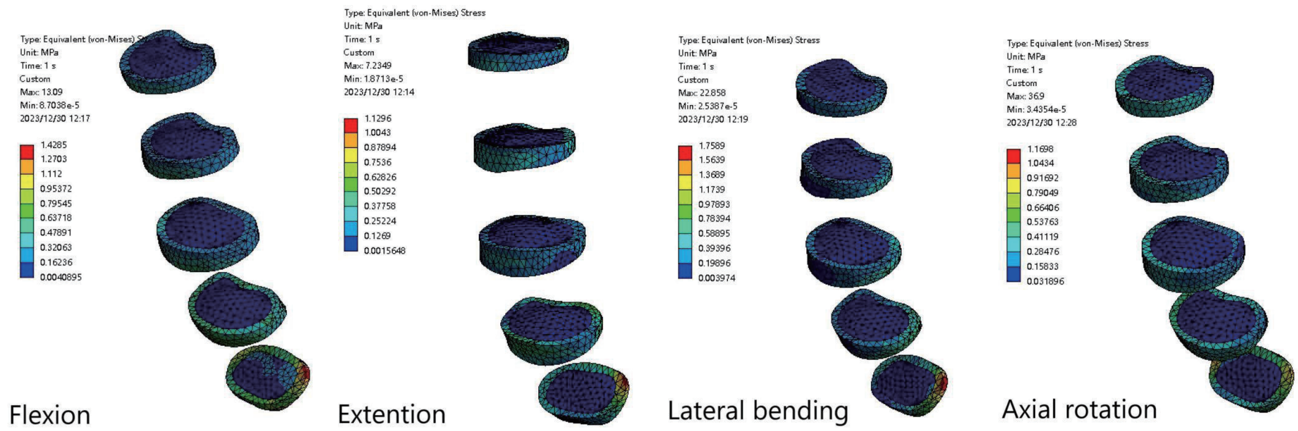

Figure 4. Stress cloud map of nucleus pulposus and annulus fibrosus

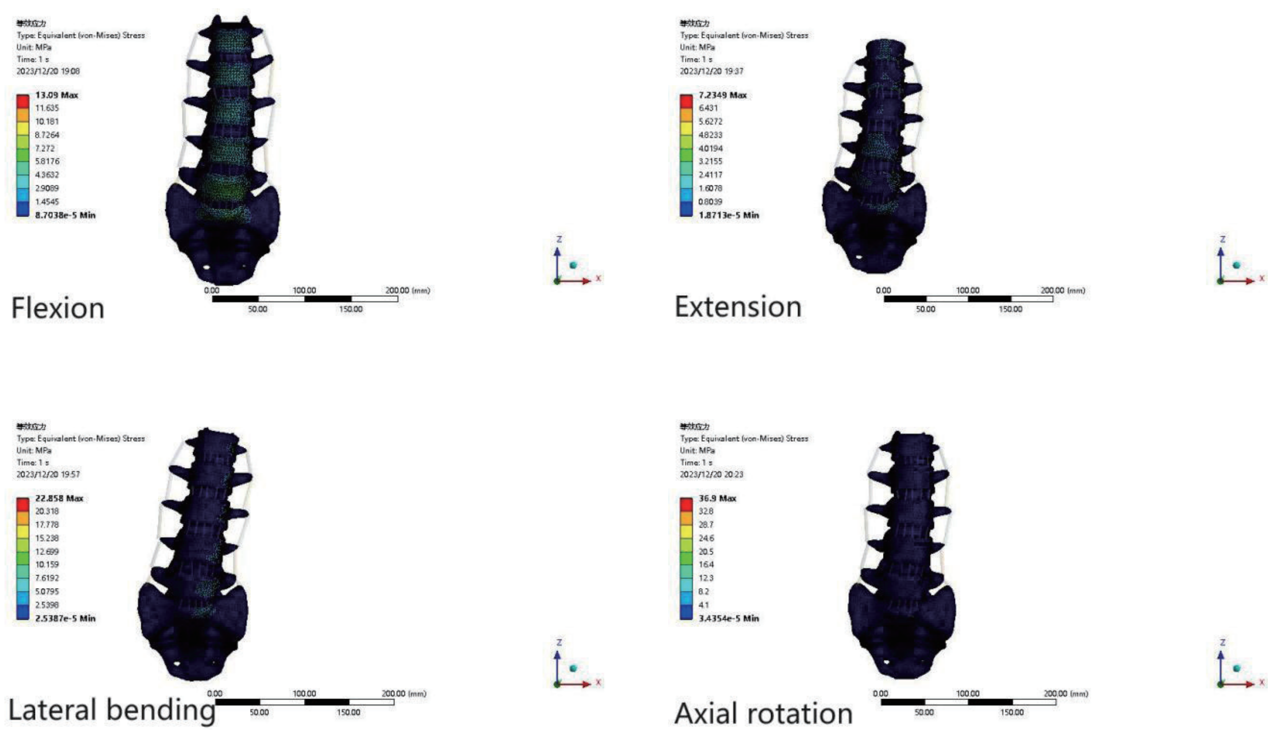

Figure 5. Front view of lumbar spine stress

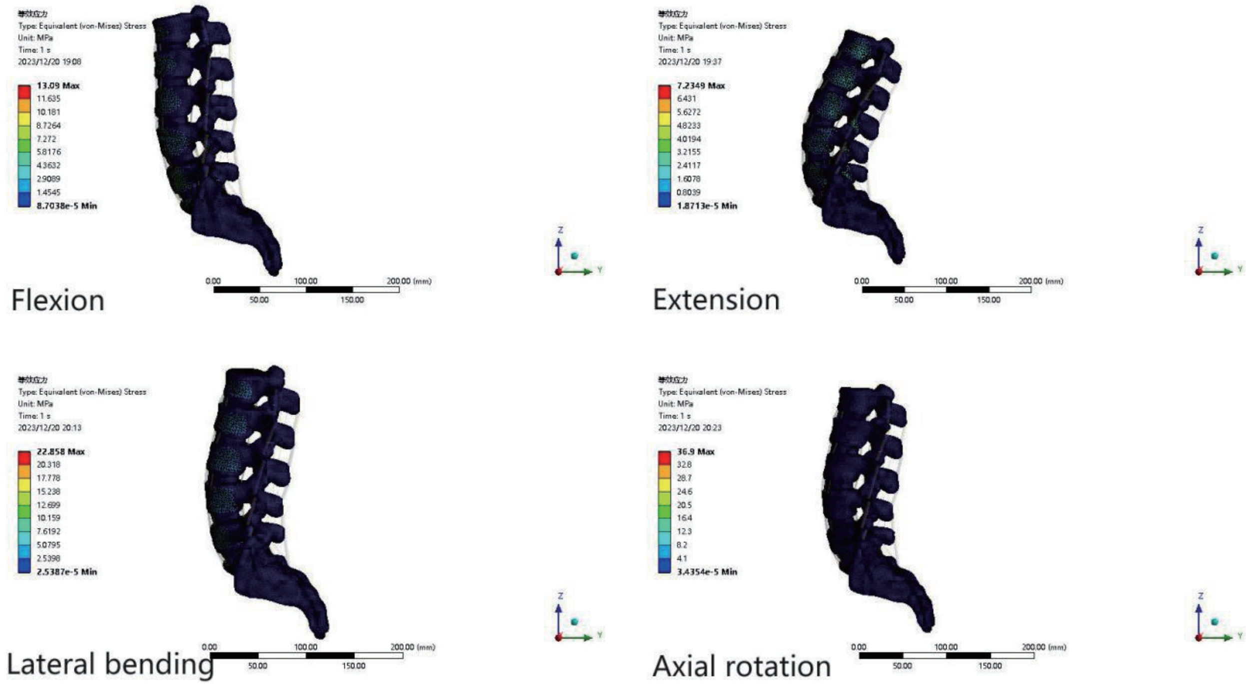

Figure 6. Side view of lumbar spine stress

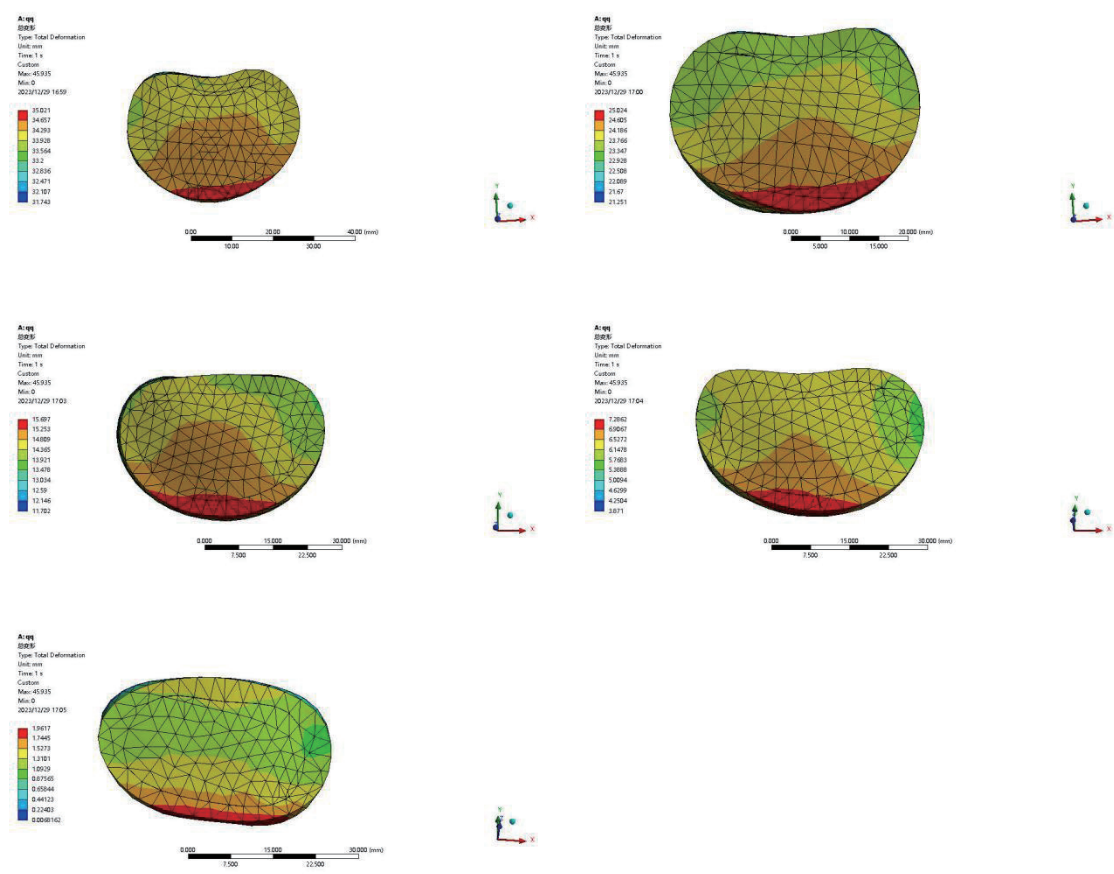

Figure 7. Cloud map of intervertebral disc displacement under forward bending condition

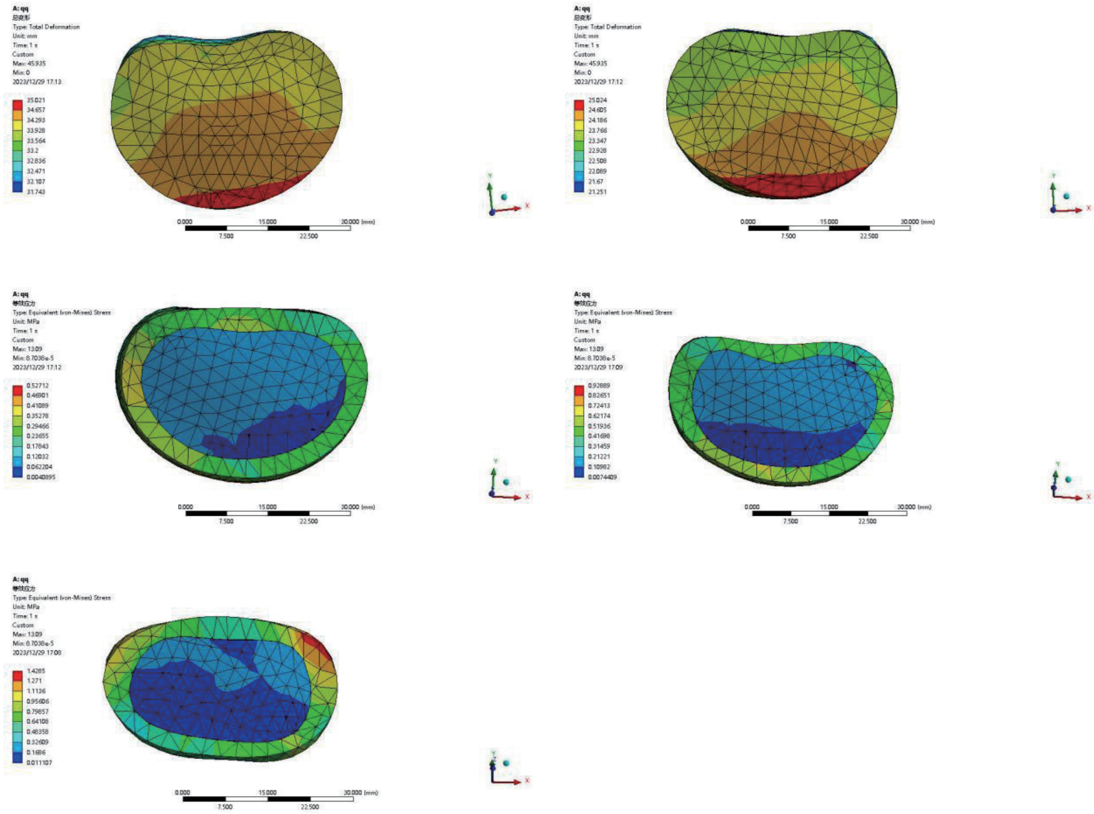

Figure 8. Stress cloud map of intervertebral disc under forward bending condition

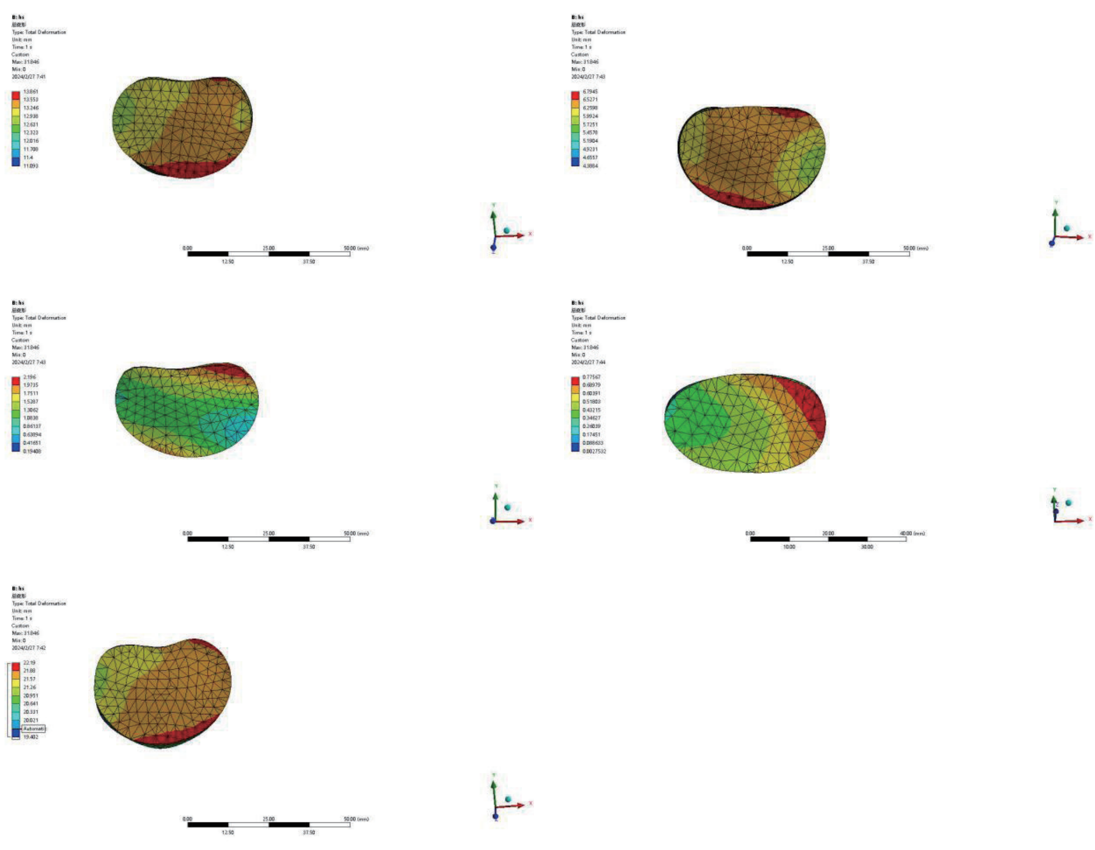

Figure 9. Cloud map of intervertebral disc displacement under backward extension condition

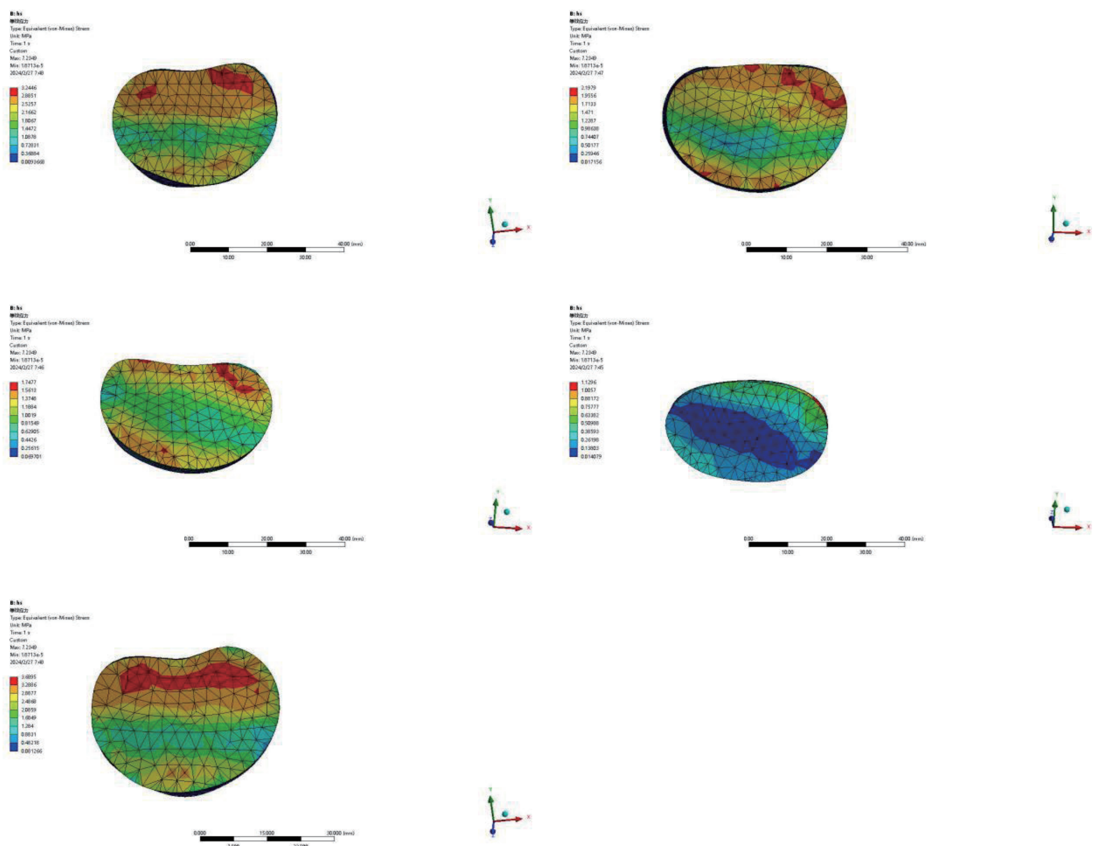

Figure 10. Stress cloud map of intervertebral disc underbackward extension condition

5. Discussion

This study establishes a normal lumbar spine model for a specific person to provide biomechanical theoretical basis for related spinal surgery. It analyzes the biomechanical characteristics of the lumbar spine model under different working conditions and modeling processes, and compares the biomechanical changes of the lumbar spine under different working conditions. Compared with relevant studies, the effectiveness of the model was verified, providing a theoretical basis for subsequent finite element research on the lumbar spine. This study is based on a lumbar CT case, constructing a human lumbar spine model using relevant modeling software, the established model has been validated by comparing the activity levels of classical literature, and is therefore considered effective. The finite element method can be used to study various difficult and complex symptoms of the lumbar spine, clinical surgical simulation, and postoperative recovery stress analysis. It was found that during the simulation of flexion in the lumbar spine model, stress was concentrated in the front of the model, and the pressure on the intervertebral disc was also concentrated in the front of the model; When the model is extended, the pressure is concentrated at the back of the model, and the stress during lateral bending and axial rotation also conforms to the in vitro experimental trend of the lumbar spine.

The finite element method has significant advantages over biological sample experiments. It can achieve experimental effects that are difficult to simulate in vivo experiments using computer technology without using patients as experimental subjects. In finite element analysis of the spine, various activity states of the spine can be simulated by changing the load size and boundary conditions. The pathological changes of the spine can be simulated by changing the appearance of the model and assigning different material properties. Given the diverse symptoms and complex etiology of spinal diseases in clinical practice, with the development of the times, finite element methods are often used in the field of spinal research to analyze the stress of implants during spinal surgery, explore the etiology, surgical efficacy, and simulate the postoperative recovery process. Finite element biomechanical analysis can establish corresponding models to simulate the biomechanical state of corresponding structures during disease progression, or establish models for treatment based on preoperative medical imaging data of patients, in order to identify causes and optimize treatment strategies.

6. Conclusion

The finite element model of the lumbar spine is a highly complex model, simplification and assumptions need to be made when building the finite element model. Due to the different material properties of different biological structures, in this study, it is assumed that trabecular bone is a homogeneous, isotropic, and linearly elastic material. The simplification of material properties may have an impact on the stress distribution of the vertebrae in the model, but it has a relatively small impact on the overall deformation of the lumbar spine and the stress distribution in the fibrous ring. The effectiveness of the finite element model was validated by comparing it with experimental data. When the spine is subjected to torque loading in various common postures in daily life, the values of spinal deformation and intervertebral disc pressure are basically consistent with the experimental values in the reference literature. This experiment established an effective healthy lumbar spine model based on CT scans of a 23-year-old healthy man. The ligaments established in this model are all fibrous ligaments, including 5 anterior longitudinal ligaments, 5 posterior longitudinal ligaments, 1 supraspinous ligament, 3 interspinous ligaments, 3 ligamentum flavum, 1 transverse process ligament, and 4 joint capsule ligaments per joint. Joint cartilage, upper and lower endplates, fibrous rings, nucleus pulposus, cortical bone, and cancellous bone were established. Through comparative in vitro experiments, validity verification, and stress analysis, it is believed that this model is suitable for finite element analysis of the lumbar spine and can provide reference for finite element modeling of the lumbar spine and even the entire spine.

References

[1] Holland, I., Fundamentals of the finite element method. Computers & Structures, 1974. 4(1): p. 3-15.

[2] Welch-Phillips, A., et al., What is finite element analysis? Clinical Spine Surgery, 2020. 33(8): p. 323-324.

[3] Benca, E., M. Amini, and D. H. Pahr, Effect of CT imaging on the accuracy of the finite element modelling in bone. European Radiology Experimental, 2020. 4: p. 1-8.

[4] Goel, V. K., et al., Finite element study of matched paired posterior disc implant and dynamic stabilizer (360 motion preservation system). SAS journal, 2007. 1(1): p. 55-62.

[5] Ayturk, U. M. and C. M. Puttlitz, Parametric convergence sensitivity and validation of a finite element model of the human lumbar spine. Computer methods in biomechanics and biomedical engineering, 2011. 14(8): p. 695-705.

[6] Lu, Y., Z. Yang, and Y. Wang, A critical review on the three-dimensional finite element modelling of the compression therapy for chronic venous insufficiency. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine, 2019. 233(11): p. 1089-1099.

[7] Zhang, Q., et al., Effect of displacement degree of distal chevron osteotomy on metatarsal stress: A finite element method. Biology, 2022. 11(1): p. 127.

[8] Kluess, D., Finite element analysis in orthopaedic biomechanics, in Finite element analysis. 2010, IntechOpen.

[9] Shirazi-Adl, A., Analysis of role of bone compliance on mechanics of a lumbar motion segment. 1994.

[10] Kiapour, A., et al., Kinematic effects of a pedicle-lengthening osteotomy for the treatment of lumbar spinal stenosis. Journal of Neurosurgery: Spine, 2012. 17(4): p. 314-320.

[11] Kiapour, A. and V. Goel, Biomechanics of a novel lumbar total motion segment preservation system: a computational and in vitro study. Bonezone, 2009. 8: p. 86-90.

[12] Goel, V. K., et al., Effects of charite artificial disc on the implanted and adjacent spinal segments mechanics using a hybrid testing protocol. Spine, 2005. 30(24): p. 2755-2764.

[13] Lin, H. -M., et al., Biomechanical comparison of the K-ROD and Dynesys dynamic spinal fixator systems–a finite element analysis. Bio-medical materials and engineering, 2013. 23(6): p. 495-505.

[14] Yamamoto, I., et al., Three-dimensional movements of the whole lumbar spine and lumbosacral joint. Spine, 1989. 14(11): p. 1256-1260.

[15] McMillan, D., et al., Stress distributions inside intervertebral discs: the validity of experimental 'stress profilometry'. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine, 1996. 210(2): p. 81-87.

[16] Atlas, S. J. and R. A. Deyo, Evaluating and managing acute low back pain in the primary care setting. Journal of general internal medicine, 2001. 16: p. 120-131.

[17] Chen, C. -S., et al., Stress analysis of the disc adjacent to interbody fusion in lumbar spine. Medical engineering & physics, 2001. 23(7): p. 485-493.

[18] Xu, M., et al. Finite element method-based analysis for effect of vibration on healthy and scoliotic spines. in International Design Engineering Technical Conferences and Computers and Information in Engineering Conference. 2016. American Society of Mechanical Engineers.

[19] Somovilla Gomez, F., et al. A proposed methodology for setting the finite element models based on healthy human intervertebral lumbar discs. in Hybrid Artificial Intelligent Systems: 11th International Conference, HAIS 2016, Seville, Spain, April 18-20, 2016, Proceedings 11. 2016. Springer.

[20] Zhang, Q., et al., Analysis of stress and stabilization in adolescent with osteoporotic idiopathic scoliosis: Finite element method. Computer Methods in Biomechanics and Biomedical Engineering, 2023. 26(1): p. 12-24.

[21] Wilke, H. -J., et al., A universal spine tester for in vitro experiments with muscle force simulation. European Spine Journal, 1994. 3: p. 91-97.

[22] Rohlmann, A., et al., Effect of an internal fixator and a bone graft on intersegmental spinal motion and intradiscal pressure in the adjacent regions. European spine journal, 2001. 10: p. 301-308.

[23] Zander, T., et al., Estimation of muscle forces in the lumbar spine during upper-body inclination. Clinical Biomechanics, 2001. 16: p. S73-S80.