Study on Fracture Healing and Biomechanics Based on Finite Element Analysis

Kai Wu [1*], Jianxiong Ma [2], Yuren Du [2]
[1]Institute of Medical Engineering and Translational Medicine, Tianjin University, Tianjin 300072, China
[2]Tianjin Hospital, Tianjin University, Tianjin 300050, China
[*]corresponding author

Keywords: Finite Element Analysis; Fracture Healing; Biomechanics

Abstract: Based on the structural characteristics of femur fractures at different stages and the CT data of human femur, a medical assembly model was established in this paper. A locking pressurized bone plate was adopted for the fixation of middle femur fractures. Finite element analysis was used to analyze the stress distribution of the locked compression plate and femur at different healing stages under 70kg physiological load and constraint conditions. The results showed that soft tissue formation and fracture healing did not occur, the stress distribution of the locking compression plate was similar to that of the femur, and the maximum stress value in the femur was higher than that of the natural femur, and the femur above the fracture was the main stress distribution area. In callus formation stage, the maximum stress of femur is lower than the ultimate strength of normal bone, and some stress is transferred downward. With proper rehabilitation training, new bone growth can be promoted.

In the treatment of fracture, the application of locking compression plate fixation is more extensive, especially in the long bone fracture site. In the working environment, the mechanical characteristics of the femur and the mechanical condition of the locking pressurized bone plate provide important references for rehabilitation training, development and design of locking pressurized bone plate, etc. At present, the mechanical characteristics of the locking compression plate are studied, with focus mainly on theoretical analysis and vitro measurement analysis. With the development of computer technology and finite element analysis, accurate mechanical characteristics can be obtained through computer data simulation technology. In this paper, a fracture treatment model was established based on the dimensions of typical locking screws, locking compression plates, and CT scan data. Finite element analysis was used to analyze the mechanical characteristics of the internal fixation system within the femur and the locking compression plate at different stages of fracture healing.

1. Materials and Methods

1.1 Modeling

A femur model was established based on CT scan data of a 25-year-old male who weighed 70kg and was 173cm tall. Through Pro/Engineering software, data was used to generate an accurate solid femur model. The model had a rake angle of 12° and a neck dry angle of 130°. The natural bending rate of femur shaft was 0.015-0.200, with a total length of 450mm.The distance between fracture ends is 1.5mm.The fracture plane is at an angle of 12° from the water plane. 6 locking screws were used in the bone plate system. The fracture surface is 282mm from the highest plane of the femoral head. Based on anatomical physiological requirements, the model was formed [1]. ANSYS software was introduced for finite element analysis. W model was used to represent the model with no healing stage at the fracture site. Y model is used to represent the soft tissue formation stage model in fracture healing. Model J represents the stage of callus development.Ti-6Al-4V（TC4）was used as the material of the bone plate system. The structure of the locking screw and the locking pressurized bone plate is characterized with 8 holes, with a thickness of 5.4mm and a total length of 160mm and width in back is 17mm.The front width is 18mm.Hole spacing: 18mm.The locking screw is 5mm in diameter and 35mm in length. Thread depth is 0.4mm, pitch of rod is 2mm, pitch of nut is 1mm.The nut height is 5.2mm.The nut diameter is 8.4mm.Crews should be appropriately simplified and rod took the form of cylinder with diameter of 5mm.The control model studied in this paper is set as a normal femur model.

1.2 Finite Element Model and Constraints

When standing on one foot, mechanical balance and stability are achieved, and loading force F exists in the femoral ankle support surface. The femur is supported by physiological loading [2-3].The same is true after locking the compression plate for internal reduction. In this study, a section was made at the top of the femoral head, perpendicular to the shaft of the force on the femur, to facilitate the loading of the model. The loading force is 700N, and the direction coincides with the force line downward. The total displacement restricts the condylar surface of the femur. Under this load, the locking platen system and the femoral strain are small. All parts of the model are treated as isotropic elastic materials and are uniformly continuous [4]. In the simulation calculation, the three-dimensional 10-node tetrahedral solid unit is used to divide the grid. The mesh at the fracture site was made in detailed manner. Binding constraint on contact surface of each component. After grid division, the number of nodes in the model is about 1.4 million, and the total number of cells is about 900,000.The calculation accuracy is good.

1.3 Safe Load

Different loads were applied to each model, J model: 2800N, 2100N, 1400N.Y model: 200N, 300N, 400N, 500N. Model W: 70N, 100N, 150N, 200N, 250N, 300N, 350N, 500N.

2. Result Analysis

Under the control of model load 700N, the maximum stress value in the femoral caltrocar region was 38.48mpa, which was lower than ultimate strength of cortical bone. The medial side of femur is the compressive stress side. The lateral side of femur is the tensile stress side. As shown in the Figure below, the mean equivalent stress distribution cloud map is compared with the backbone segment of the model. The stress was concentrated on the longitudinal ridge behind the backbone, and the average equivalent stress was 0.04-19.76 MPa.

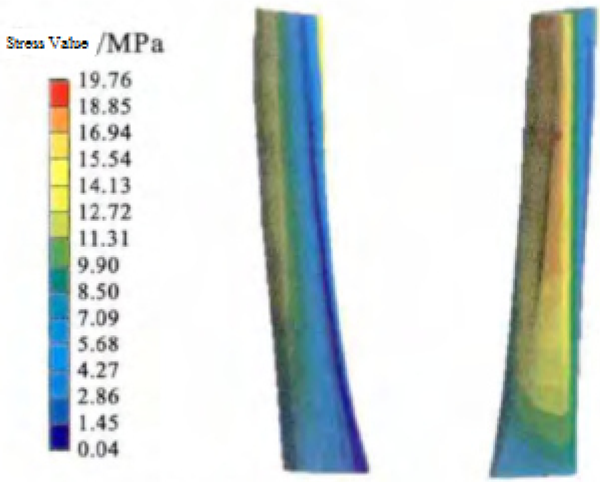

Figure 1. Stress distribution nephogram of the femoral shaft of the control model

During the treatment of femur fracture, the stress distribution Nephogram of the locking compression plate is shown in Figure 2 below. Under the action of load 700N, in the W model, stress concentration of the locking pressurized bone plate appears in the areas between the thread holes of the bone plate and the contact surface of the nuts of the third and fourth locking screws, between the fifth and sixth thread holes, and between the third and fourth thread holes, with a maximum of 1,796.32mpa.At the interface between the first locking screw and the medullary cavity of the medial femur, the maximum equivalent stress value in the femur was 375.36mpa, which was higher than the ultimate strength of cortical bone. The contact surface between the first, second, and third screws and the femoral bone marrow cavity is the stress concentration area in the femur. For the Y model, the stress concentration of the locking pressurized bone plate appears near the thread holes of the bone plate and the contact surface of the nuts of the third and fourth locking screws, between the fifth and sixth thread holes and between the third and fourth thread holes, with the maximum stress value of 1561.23mpa.Stress concentration occurs in the contact surface between the bone plate and the lateral femur, and between the first, second, and third screws and the femoral bone marrow cavity. The contact surface between the nail rod of the second locking screw and the medullary cavity of the medial femur showed the maximum stress in the femur, 275.89mpa.For Model J, the maximum stress value of 205.17mpa appears at the separation interface between the locking screw nut and the right side of the second threaded hole on the front of the bone plate. Stress concentration occurs between the second to the sixth composite hole on the right side. Stress concentration in femur occurred near the contact surface of the sixth locking screw and the lateral contact surface of the shaft above the contact surface of the second locking screw. The first locking screw was in contact with the lateral femur, and the maximum stress value within the femur was 70.13mpa.

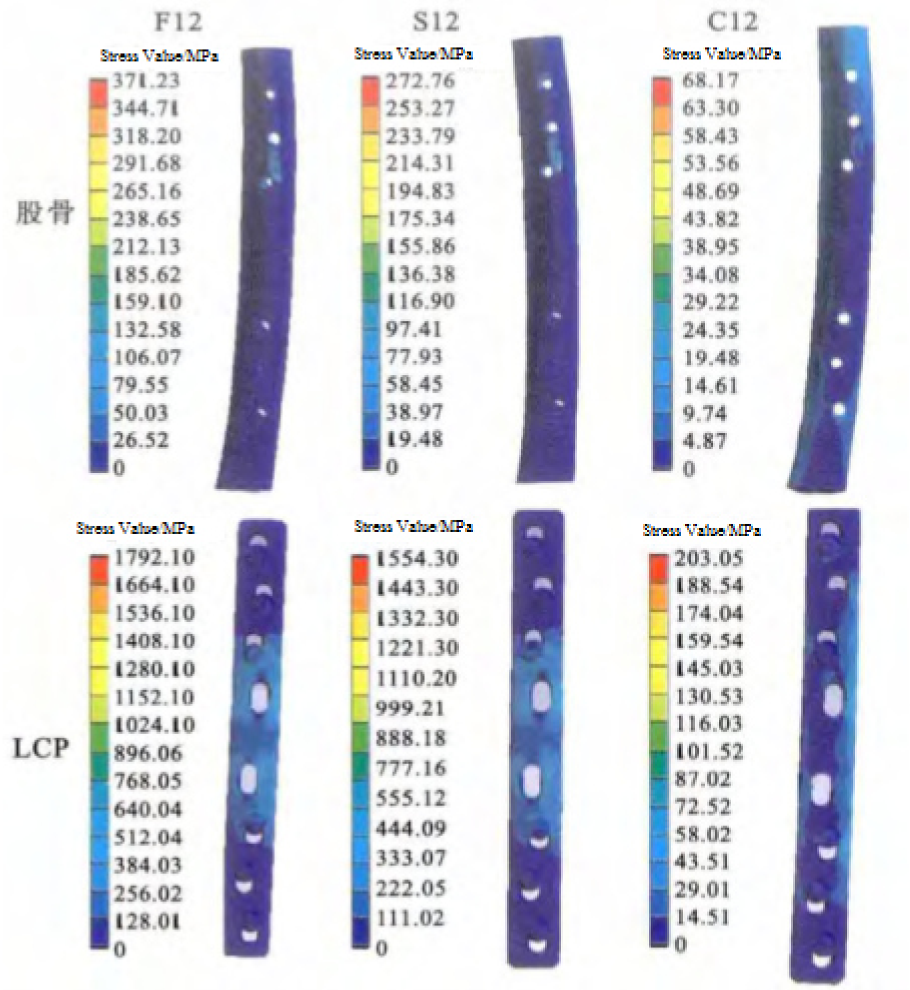

Figure 2. Nephogram of stress distribution in different stages of the model of locking compression plate

Compared with the control model, the maximum stress of femur in Y model and W model was significantly higher. Although the maximum stress in the femur of model J is also relatively high, it is within the safe range. Comparing the ultimate strength and maximum stress of various materials, the safety load of Y model is 376N, and that of W model is 325N.Therefore, in the stage of Model Y and model W, the load in patients' recovery training needs to be lighter. The safety load of Model J is 2891N. In model J, it is appropriate to get out of bed and carry out rehabilitation training.

3. Discussion

In this paper, a model was established based on the CT scan data of femur. The number of nodes and elements in the finite element model is more than one million with high precision. According to the physiological load of normal physiological behavior of human body, 700N load is selected, and the constraint conditions tend to be reasonable. With the continuous research on intra - bone plants, the production requirements of implant materials are also higher [5-6].The most important factors are mechanical compatibility and biocompatibility. Mechanical compatibility includes wearability, fatigue strength, toughness, stiffness and so on. The strength and stiffness of the material are related to the strength and stiffness of the implant. The materials of the pressurized bone plate and screws can be locked. Pure titanium, titanium alloy, stainless steel, composite materials, etc., have good biocompatibility [7-8].And ductility, strength, stiffness show good performance. But more important is the influence of spatial structure and shape on mechanical compatibility. This study shows that in the W model stage, the internal bracket role of the internal fixation system of the pressurized bone plate is locked to provide stress for the upper part of the broken bone. The internal fixation system and the locking screw at the fracture site are under unilateral stress, so maximum stress occurs on the locking compression plate. In the Y model stage, the maximum equivalent stress value in the femur and the maximum equivalent stress value of the locking compression plate were significantly reduced compared with those in the W model stage. Due to the formation of soft tissue, the stress on one side is weakened. In model J, the maximum equivalent stress was significantly reduced, and the maximum equivalent stress still appeared on the locking compression plate. As the elastic modulus of the material increases, the stress of the medial side of the bone is transferred to the femur below the fracture site, mainly in form of callus [9-10].

4. Conclusion

In this paper, the biomechanics of fracture healing at different stages were studied to provide a mechanical reference for the design and improvement of the locking compression plate and its shape and spatial structure. It is of certain application value to calculate the safety load that can be borne at each stage, so as to provide help in making the clinical rehabilitation training plan.

References

[1] Lin Juanying, Liu Xiaoying, Xing Lijie, et al.Biomechanical analysis of calcaneal bone based on finite element method. Journal of Medical Biomechanics, 2008,33(1):37-41.

[2] Zhang Haowei, Chen Liang, Yang Junyan, et al.Biomechanical study of heel pain in push-off stage based on finite element method.Chinese Journal of Biomedical Engineering ,2020,39(2):190-196.

[3] Ding Kai, Chen Wei, Yang Weijie, et al.Finite element analysis of the effect of residual internal/external varus malunion after femoral fracture on biomechanics of knee.Chinese Journal of Geriatric Orthopaedics & Rehabilitation (Electronic Edition),202,6(5):257-264.

[4] Xu Duo, Sun Ruijian, Wang Tianyu, etc.Exploration and reconsideration of distal femur Type A fracture based on finite element biomechanical analysis.Chinese Journal of Surgery|,2019,57(11):812-817.

[5] Azguli Karim, Uzhkaxi Aiyiti, Teng Yong.Finite element analysis of femoral shaft fractures fixed with bone plate.Mechanical Design and Manufacturing,2020,(9):172-175.

[6] Zhang Xinyuan, Ding Xiaohong, Duan Pengyun.A finite element model of hip joint based on the distribution characteristics of bone material and its stress analysis.Beijing Biomedical Engineering ,202,39(5):462-469.

[7]Xia Changjiang, Yuan Zhifeng, Fang Ning. (in Chinese)Analysis of biomechanical characteristics of distal radial fractures based on ulnar and radial three-dimensional finite element model.Chinese Journal of Tissue Engineering Research ,202,24(6):893-897.

[8] Li Yongyao, Zhao Yong, Cheng Hao, etc.Distal radius Frykman Ⅷ type plate fixation fracture finite element model, and mechanics analysis.Chinese Journal of Tissue Engineering Research ,202,24(27):4272-4277.

[9] He Xiaoyu, Wang Chaoqiang, Zhou Zhiping, etc.Modeling and biomechanical analysis of healthy foot bones and common diseases by three-dimensional finite element method.Chinese Journal of Tissue Engineering Research ,2020,24(9):1410-1415.

[10] Ma Pengpeng, Cai Ming, Li Wei, et al.Biomechanical analysis of vertebroplasty bone cement hardness by three-dimensional finite element modeling.China Digital Medicine ,2020,15(6):93-96.