International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2018): 7.426

Research and Development of Combined External Fixation based on Finite Element Analysis

Peng Du

Tianjin Jingdu Beisida Biotechnology Co., Ltd., Xiditou Town, Beichen District, Tianjin 300222, China
396628749[at]qq.com

Abstract: ***There are many types of external fixation brackets for medical devices, and there are 20 kinds of lower limbs. Each type of product has different specifications according to size. Therefore, it is especially important to choose an external fixation bracket that conforms to the best state of human bones. With the development of computer simulation technology, ANSYS Workbench software has been widely used in many fields. Through the static analysis of ANSYS Workbench finite element analysis software in the development of external fixed bracket products, the efficiency of new product structure evaluation and structure optimization can be effectively improved. And it can be infinitely close to the force of the stent under the real state and the ability of the human bone to withstand, avoiding repeated experiments, reducing the experiment cost and shortening the cycle of R&D registration.***

Keywords: ANSYS Workbench Finite Element Analysis Combined External Fixation

1. Introduction

With the development of external fixation technology, combined external fixation stent has become one of the more important tools in the minimally invasive treatment of fractures. Many orthopedic companies are actively developing the combined external fixation products. Under such pressure, how to quickly develop new products, shorten the research and development cycle, and seize the market is particularly important.

2. Product Introduction

The use of a combined external fixation stent to treat the fracture is to properly reset the affected limb and restore function as soon as possible. The method is to insert different bracket accessories such as long spiral nails and steel needles into the fracture ends, and then add connecting rods, half rings and steel needle fixing clips. The fracture is brought to anatomic reduction (or functional reset) and relatively fixed purpose.

The combined bone external fixator is made of aluminum alloy, stainless steel and titanium alloy. Most of the parts can be interchanged and combined. When used, it can be composed of different configurations according to the fracture site and fracture type. It can be composed of unilateral and bilateral. The configuration of the type, the semi-ring type, the frame type, or the like may be arbitrarily formed into various forms of the external fixator according to the needs of the fracture.

3. Theoretical analysis and working principle of external fixed bracket

The CAE software linear static analysis system was used to analyze the external fixed support, and the pressure of the bone under the real state was simulated to determine the type and size of the best external fixed support. In order to verify the validity of the established 3D model, the established 3D model was imported into ANSYS for finite element analysis, material properties were defined, appropriate boundary conditions and contact types were established, and the analysis results were obtained.

4. Work Content

4.1 Experimental analysis result

According to the previous three experimental data, the changes of torsion and four-point bending are summarized, and then compared with CAE results to determine the accuracy of CAE analysis.

4.2 Reverse engineering modeling

To increase the design function, firstly, the product point cloud is collected by the non-contact scanner, and the point cloud is processed by the reverse software Geomagic, noise reduction, encapsulation, surface fitting, and product modeling. Compare the maximum modeling error and optimize it by comparing the model with the real object.

4.3 Establish database

Establish a complete product model database for external fixation of lower extremities, and list different models of different specifications.

4.4 CAE analysis

Calculate the CAE computer analysis software, load the bone and the external fixed bracket, define the material properties respectively, assign the corresponding materials to each component, then set the contact between the components, then divide the mesh, and then set the boundary conditions --- Add constraints and loads, and finally click Solve to get the results and analyze them. The torsion test and the four-point bending test were respectively performed to calculate the actual bearing capacity of the bone.

4.5 Result output

The actual bearing capacity of each part of the bone is viewed through a chromatogram, and the optimal product type and size are determined according to national standards, and the cycle of design and analysis is shortened. The maximum and minimum values of each variable are marked in the following figures. It can be seen from the figure that the maximum values of the total deformation and the total stress occur near the load loading, which is consistent with the actual situation and the theoretical value. The experimental values, axial compression test and the calculated values of the finite elements are compared below. As shown Figure 1, 2, 3.

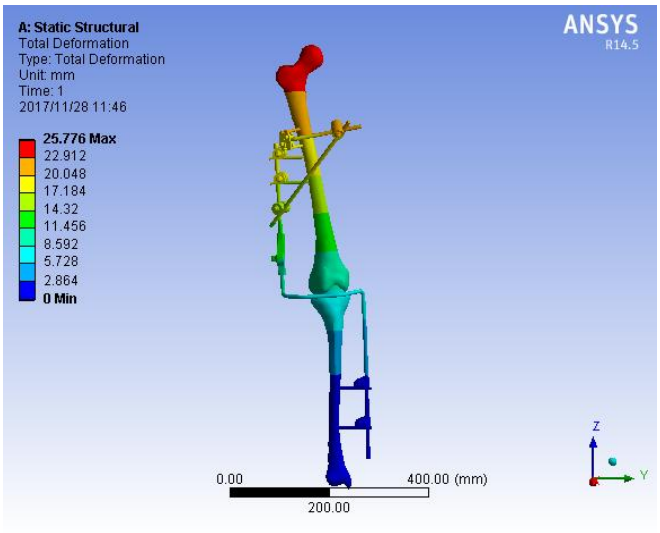

Figure 1: Analysis of axial compression test

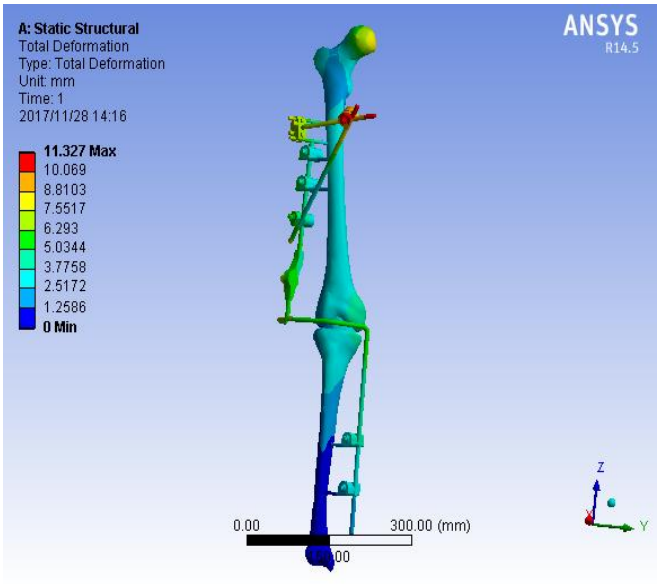

Figure 2: Torsional analysis

Table 1: Torsion test value and finite element comparison

Test and simulation	Test 1	Test 2	Test 3	finite element
Corner (°)	11.68	11.35	11.54	11.32

It can be seen from the comparison of the above table that the calculated value of the finite element is close to the experimental value, so the model has certain reference value.

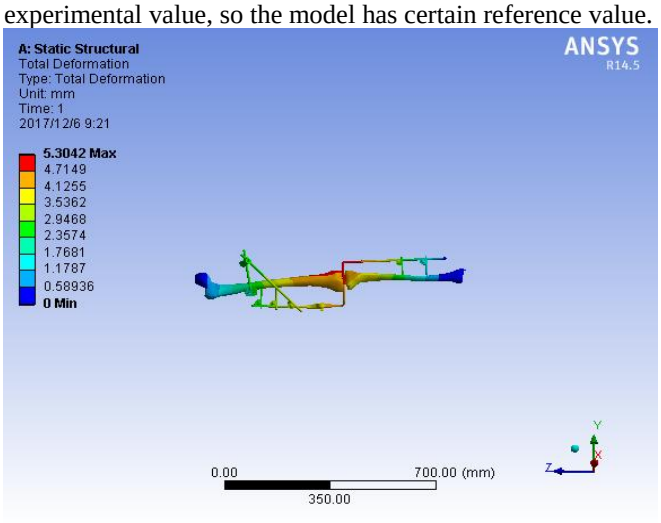

Figure 3: Torsional analysis

Table 2: Four-point bending experiment analysis

Test and simulation	Test 1	Test 2	Test 3	finite element
Displacement (mm)	5.22	5.47	5.36	5.30

The test deformation value and the finite element analysis value are compared in the table, and the test value is taken as the displacement value under the load of 1000N. Analysis and comparison show that the finite element calculated value is close to the experimental value.

4.6 CAE curve

The simulation verification curve in CAE software is used to verify the deformation of the same type of product and shorten the experimental period. As shown Figure 3.

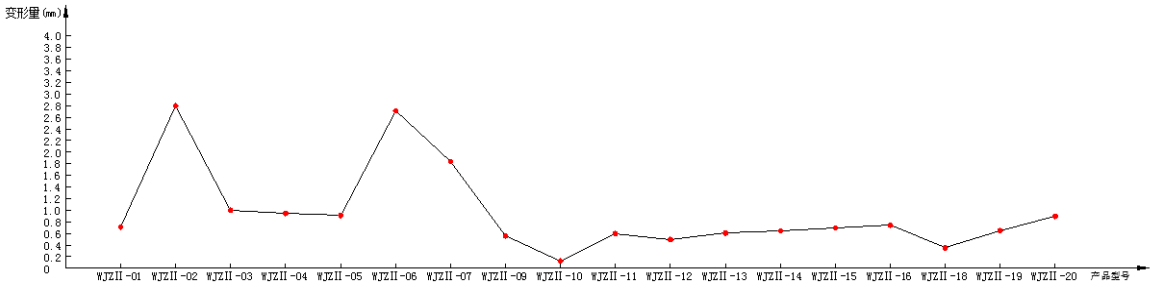

Figure 3: Simulation verification graph

5. Equations

In the past, product development rarely used finite element analysis software for simulation analysis, which greatly increased the cycle and cost of the company's new product development. Through CAE analysis, it can accurately and quickly determine the best external fixation bracket for bones, shortening the development cycle and reducing the experimental cost.

6. Experience sharing

How can enterprises seize the market in the rapidly developing orthopedic market? First, we should introduce advanced R&D design concepts and methods to continuously learn and think about how to apply them to the technology of our own enterprises.

References

[1] Ozgur Verim, M. Volkan Yaprakci, Abdurrahman Karabulut, “Biomechanical Optimization of a Novel Circular External Fixator (Optimization of Circular External Fixator),” Springer (37), pp.760~768, 2017.
[2] C. Li, “Mechanical performance and finite element analysis of external fixator for adjustable pressure Department of orthopedics,” School of mechanics and Engineering Science, Zhengzhou, 2018.
[3] Be'ery-Lipperman M, Gefen A, “A method of quantification of stress shielding in the proximal femur using hierarchical computational modeling,” Comput Methods Biomech Biomed Engin, pp.35~44, 2006,9(1).

Author Profile

Peng Du is a product development engineer at Tianjin Jingdu Beisida Biotechnology Co., Ltd., Xiditou Town, Beichen District, Tianjin 300222, China