A Novel Finite Element Method based Biomechanical Model for HIT-Robot Assisted Orthopedic Surgery System

Jia Zhiheng, Du Zhijiang, Wang Monan
Robotics Institute, Harbin Institute of Technology, Harbin, China

***Abstract*—To build a biomechanical human model can make much sense for surgical training and surgical rehearse. Especially, it will be more meaningful to develop a biomechanical model to guide the control strategy for the medical robots in HIT-Robot Assisted Orthopedic Surgery System (HIT-RAOS). In this paper, based the successful work of others, a novel reliable finite element method based biomechanical model for HIT-RAOS was developed to simulate the force needed in reposition procedure. Geometrical model was obtained from 3D reconstruction from CT images of a just died man. Using this boundary information, the finite element model of the leg including part of femur, broken upper tibia, broken lower tibia, talus, calcaneus, Kirschner nail, muscles and other soft tissues was created in ANSYS. Furthermore, as it was too difficult to reconstruct the accurate geometry model from CT images, a new simplified muscle model was presented. The bony structures and tendons were defined as linearly elastic, while soft tissues and muscle fibers were assumed to be hyper elastic. To validate this model, the same dead man was involved to simulate the patient, and a set of data of the force needed to separate the two broken bones and the distance between them in reposition procedure was recorded. Then, another set of data was acquired from the finite element analysis. After comparison, the two sets of data matched well. The Finite Element model was proved to be acceptable.**

Keywords—**Finite Element Method, Biomechanical model, Hyper elastic, Orthopedic Surgery**

I. INTRODUCTION

Orthopedic surgery is one of the most common operations in hospitals, but precise reposition of broken bones, the major procedure before fixing in fracture surgery [2], is always a challenge to most surgeons, sometimes even to experienced experts [1]. The dragging is the popular antagonistic action on the beginning of reposition, which is used to deal with retraction and imbedding. The strength of traction should be gradually, continuously increased to avoid steep rising. The demand for pulling manner, pulling strength and distance is varied with breaking position. [3]. So it can make much sense to develop an anatomically realistic biomechanical model for surgical training and surgical rehearse. Especially, it will be more meaningful in HIT-RAOS, as it can guide the control strategy for the reposition parallel robot.

However, surgical simulation of human bodies is an extremely demanding application of deformable modeling where large deformation occurs. Over the past decades, there are two principle approaches for biomechanical modeling: the mass-spring approach, and Finite Element Method (FEM). Nevertheless, mass-spring models [4] comprise a set of nodes connected by springs, with point masses attached at each node. Real time performance can be achieved with a limited number of nodes, but the behavior is often unrealistic and can be unstable.

The FEM approach is a little slow, but produces more accurate results [5], since the true elastic behaviors of soft tissue are nonlinear. The FEM is good at biomechanical analysis, and has been proved to be a promising tool to investigate the internal information inside humans.

Furthermore, though many successful works, such as [6] [7] [11] have been done using FEM to explore the biomechanical characters of lumbar spine, foot, arm and so on, few researchers focused on legs, especially for the orthopedic surgery. And seldom actual experiments were conducted to validate their models in orthopedic surgery.

This objective of this paper was to develop a comprehensive FE model of leg under anatomical considerations, which was built from the actual 3D geometry model, to investigate and simulate the broken leg's biomechanical characters during the orthopedic surgery. At last, the FE model was certified by real experiments.

II. METHODOLOGY

A. Geometric modeling

The geometrical models were obtained from 3D reconstruction from CT images of the left leg of just died normal subject (male, age 30, height 170cm and weight 65kg), which was donated by Harbin Medical University the 2nd Affiliated Hospital . The CT images were taken with intervals of 1mm in the neutral position. The images were manual segmented with the help of MIMICS v8.10 (Materialise, Leuven, Belgiu). Then the boundaries of skeleton and skin surface were acquired using 3D reconstruction algorithm, and outputted as STL format, which can be seen in Fig.1a. Additionally, the geometry model of an assistant device, Kirschner nail, was built by CAD software according to its real dimension.

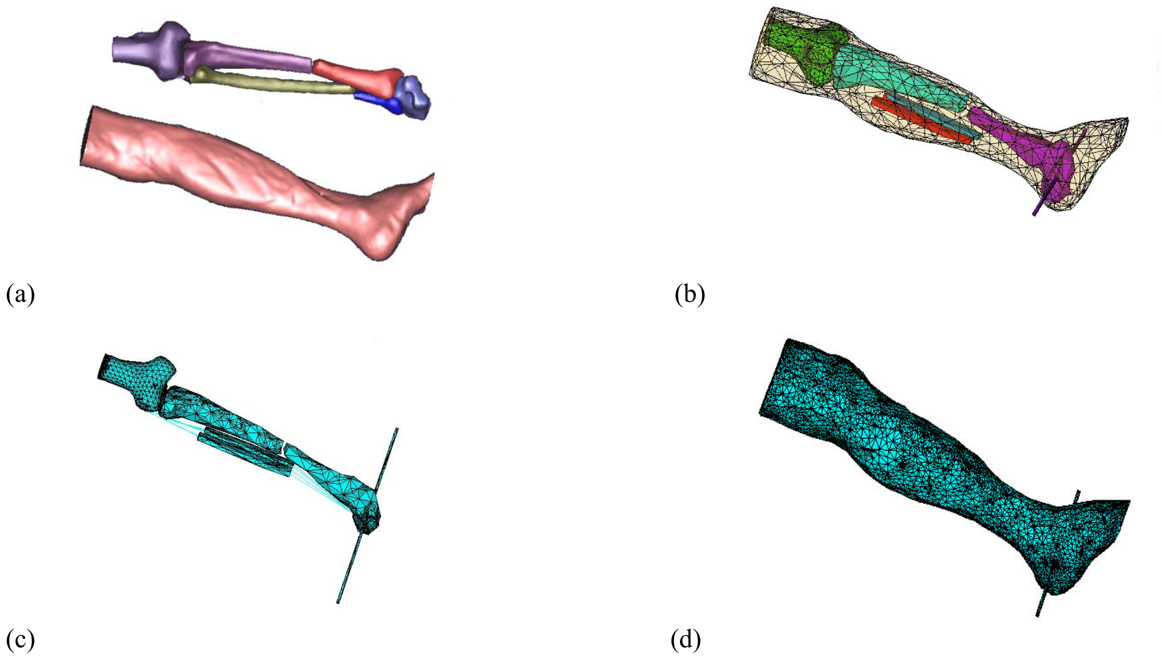

Fig. 1 (a) Surface model created from CT images. (b) Solid model created in ANSYS. (c) Muscle and bone models. (d) Meshed model in ANSYS.

TABLE I
Material properties and element types of the finite element model

Component	Element type	Young's modulus E (MPa)	Poisson's ratio v	Cross-sectional area (mm^2)
Bone	Solid185	7300	0.3	
Muscle	Solid185	Mooney-Rivlin Hyper elastic model		
Soft tissue	Solid185	Polynomial Form Hyper elastic model		
Kirschner nail	Solid185	21000	0.3	
Tendon	Link10	350		290.7

B. Finite element modeling and materials

The models, as shown in Fig1b, c and d, were created by ANSYS v5.7 (Swanson Analysis System Inc., Houston, TX). The STL files produced by 3D reconstruction contained the boundary information of geometrical models, which were made of a series of triangles. Trough some lexical analysis, we could get to know the point position and point sequence which make up of the model surface. Then in ANSYS, ANSYS Parametric Design Language (APDL), a powerful language for optimizing the FEM workflow, was employed to create the solid models for each bone and tissue. Firstly, key points were created. Secondly, triangles, which make up the surface of models, were formed through known point sequences. Last, solid models were produced from the closed surface, which can be seen in Fig1b. After all the models were finished, they were then assembled together according to their real position. Additionally, the model of an assistant device, Kirschner nail, has also been built and added to the FE model.

Totally, the FE model consisted of part of femur, broken upper tibia, broken lower tibia, talus, calcaneus, Kirschner nail, muscles and other soft tissues. The bones and muscles were all embedded in a volume of soft tissues, which included fat, and other tissues whose geometry were too complicated to be reconstructed. The bony structures and tendons were defined as linearly elastic, while the soft tissue and muscle fibers were assumed to be hyper elastic.

According to the model developed by Gefen et al [8], the Young's modulus and Poisson's ratio for bony structures were assigned as 7300 MPa and 0.3, respectively. This value was selected by weighing cortical and trabecular elasticity values.

Depending on anatomical knowledge, gastrocnemius and musculus soleus are the two main muscles in the reposition procedure, so other muscles were ignored. Furthermore, as it was too difficult to extract the accurate contour of muscles from CT images, a new simplified muscle model was adopted under surgeon's suggestion. The muscle was divided into two parts: tendon and muscle fibers. The tendon was regarded as tension only truss, and the muscle fiber was defined to be Mooney-Rivlin, which was applied for the elasticity potential function W of incompressible hyper elastic body [9].

Considering the computing efficiency, two-parameter Mooney-Rivlin model was used in this paper. And its function used in ANSYS is listed as follows [11]:

$$W = C_{10}(\overline{I}_1 - 3) + C_{01}(\overline{I}_2 - 3) + \frac{1}{d}(J-1)^2 \qquad (1)$$

Where: W is strain energy potential. $\overline{I}_1$ and $\overline{I}_2$ are the first and second deviatoric strain invariants defined as

$$\overline{I}_1 = \overline{\lambda}_1^{\,2} + \overline{\lambda}_2^{\,2} + \overline{\lambda}_3^{\,2} \qquad (2)$$

$$\overline{I}_1 = \overline{\lambda}_1^{\,-2} + \overline{\lambda}_2^{\,-2} + \overline{\lambda}_3^{\,-2} \qquad (3)$$

With the deviatoric stretches $\overline{\lambda}_i = j^{-1/3}\lambda_i$. J and λ_i are the elastic volume ratio and the principal stretches, respectively. C_{10} , C_{01} are the material constant characterizing the deviatoric deformation of the material. d is the material incompressibility parameter. J. Teran's work [12] has revealed typical values for these parameters:

C_{10}=30000Pa, C_{01}=10000Pa, and K=60000Pa. As the initial bulk modulus is defined as:

$$K = \frac{2}{d} \qquad (4)$$

So we could get d =1.667×10^{-5}Pa^{-1}.

Another hyper elastic material model, Polynomial Form, was used to represent the nonlinear characters of the encapsulated soft tissue. The following is its second-order function:

$$W = \sum_{i+j=1}^{N} C_{ij}(\overline{I}_1 - 3)^i(\overline{I}_2 - 3)^j + \sum_{k=1}^{N} \frac{1}{d_k}(J-1)^{2k}$$

Where C_{ij} and d_k are material parameters, their typical values can be found in [7]; The definition of W , J , $\overline{I}_1$ and $\overline{I}_2$ are the same as them in Mooney-Rivlin model. All the material properties and element types in the FE model could be found in Table.1.

C. Load defining

In the reposition procedure of orthopedic surgery, the main work is to separate the two broken bones and align them. As the Fig.2 shows, the upper cru was fixed to a bracket using a belt. And the lower cru was fixed to the reposition parallel robot through a Kirschner nail, which drilled though the calcaneus, then was fixed to the parallel robot.

Fig. 2 The load conditions in orthopedic surgery

As Fig.3 shows that, according to the real conditions in the orthopedic surgery, six degrees of freedom of the points besides the belt in the upper crus were defined as zero. In the two sides of the FE model of Kirschner nail, two forces were applied. Their value was the half of the parallel robot used to separate the two broken bones. Their directions were the same as that of parallel robot moved.

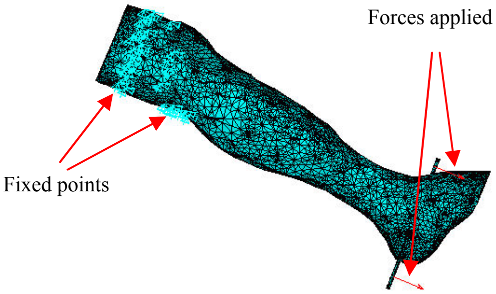

Fig. 3 The load conditions in Finite Element Model

III. EXPERIMENT

To validate the authenticity of the FEM based biomechanical model for HIT-RAOS, a reposition experiment was conducted. The force needed to separate two broken bones and the distance between them was recorded. While another set of data could be acquired from the simulation using finite element models we have built.

In the experiment, the dead man, who was the same as the one being scanned to obtain CT images, was involved to simulate the patient in real surgery. From the unloaded situation, every 10s the parallel robot moved 1 mm forward. At the same time the force needed could be acquired from the force sensor, which was placed between the parallel robot and the foot of the patient as Fig.4 shows. What's more, because there might be some displacement produced in the fixed upper crus, bended Kirschner nail or the cartilage between talus and calcaneus, it was not accurate to regard the distance parallel robot moved as the distance between the broken bones. In fact, we employed X-ray images to calculate the distance between the bones, which were taken every time when the force was read from force sensor. And a ruler embedded with steel balls in fixed distance was used for calibration.

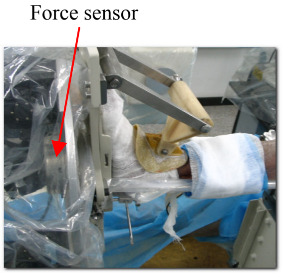

Fig.4 The force sensor used to record the force needed to separate two brokent bones

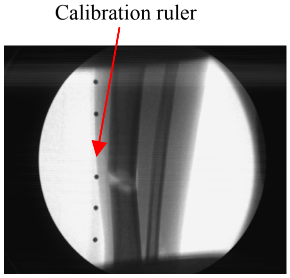

Fig. 5 The X-ray image used to calculate the distance between two broken bones

As Fig.6 shows, the solid line is the data came from the experiment, and the doted line represents the result from finite element simulation. The x axis is the distance between the two broken bones acquired from X-ray images. While the y axis is the force used to separate the two broken bones got form the force sensor.

Although the errors in experiment and the radical simplifications of the biomechanical model has inevitably caused the difference between the two sets of data, we can still easily reach the conclusion that the two sets of data match well in the values and trends at the range of 4~9mm, which has covered most of the distance range during the orthopedic surgery.

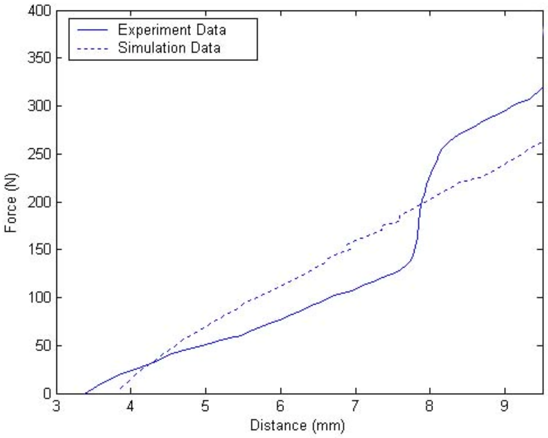

Fig. 6 The two sets of data from experiment and FEA

IV. CONCLUSION

This paper presented a novel reliable finite element model for the HIT-RAOS, which could predict and simulate the force needed in the reposition procedure. The geometrical model was obtained from 3D reconstruction from the CT images. Using this boundary information, the finite element model was created in ANSYS. The bony and ligamentous structures were defined as linearly elastic, while the soft tissue and muscles were assumed to be hyper elastic. To validate this model, a cadaver was involved to simulate the patient to get a set of data of the force needed to separate the two bones and the distance between them. Then, another set of data was acquired form the finite element analysis. The results were content and showed that the FE model was reliable.

ACKNOWLEDGMENT

The authors would like to thank Dr. DM Wu, FF Zhang, LT Yu and Master DH. Yang and etc, for their excellent work in the system. Thanks to Doctor L. Shao and C.G. Hao for their effective suggestion on the simplification of FE modeling. And especially thanks for the financial support of the National High Technology Research and Development Program of China (863 Program) (grant #2004AA421030), and National Natural Science Foundation of China (grant #60505016)

REFERENCES

[1] Gueziec A, Wu K, “Providing visual information to validate 2-D to 3-D registration”, *Medical Image Analysis*, vol. 4, pp. 357–374, 2000

[2] Kühnapfel U., Cakmak H. K. and Maass H: “Endoscopic Surgery Training using Virtual Reality and deformable Tissue Simulation”. *Computers & Graphics* 24(2000) 671-682, Elsevier (2000)

[3] M. Shoham, M. Burman, E. Zehavi, L. Joskowicz, E. Batkilin,Y. Kunicher, "Bone-mounted miniature robot for surgical procedures: Concept and clinical applications," *Robotics and Automation, IEEE Transactions on* , Vol.19, pp. 893-901, 2003.

[4] Minxiu Kong, Zhijiang Du, Lining Sun, Lixin Fu, Zhiheng Jia, Dongmei Wu " A Robot-Assisted Orthopedic Telesurgery System" *IEEE Engineering in Medicine and Biology 27th Annual Conference-Proceedings*, p 1661-1667. Shanghai, China, September 1-4, 2005

[5] Morten Bro-Nielson, Finite Element Modeling in Surgery Simulation. *Proceedings of the IEEE*. Vol. 86. No. 3. March 1998.

[6] F.Nabhani, M.Wake: “Computer modeling and stress analysis of the lumbar spine” *Journal of Materials Processing Technology* 127(2002)40-47

[7] Jason Tak-man Cheung, Ming Zhang, Aaron Kam-lun Leung, Yu-bo Fan: “Three-dimensional finite element analysis of the foot during standing---a material sensitivity study” *Journal of Biomechanics* 38(2005) 1045-1054

[9] Gefen, A., Megido-Ravid, M., Itzchak, Y., Arcan, M.:”Biomechanical analysis of the three-dimensional foot structure during gait: a basic tool for clinical applications” *Journal of biomechanical engineering,* 122, 630-639, 2000

[10] C. W. Macosco Rheology: “principles, measurement and applications”, *VCH Publishers*, 1994, ISBN 1-56081-579-5

[11] ANSYS Release 5.7 Documentation. ANSYS, Inc. Canonsburg, PA, USA

[12] Teran J, Sifakis E, Blemker SS, Ng-Thow-Hing V, Lau C, Fedkiw R.: “Creating and simulating skeletal muscle from the visible human data set” *IEEE Trans Vis Comput Graph.* 2005 May-Jun;11(3):317-28