Surgical planning tool with biomechanical simulation

B. Erdőhelyi, E. Varga, A. Kuba
University of Szeged, Hungary

Abstract The fixation of fractured often needs a very careful decision making. The operation has to be designed very carefully. A computer assisted system can help the surgeon in the planning phase to increase surgical accuracy. This paper introduces a software tool to plan a surgical intervention and to calculate the biomechanical stability of the plan. The proposed system provides 3D movement and rotation of the bone fragments and the insertion of fixation screws and plates. Finite element analysis is used to calculate mechanical stability of the surgical plan. Using these results the surgeon is able to see the week points of the fixation before the surgery. He can even try several surgical plans to pick the most promising one.

Keywords Surgical planning · Computer aided surgery · Finite element analysis

1. Introduction

By using CAS systems the treatment can have several advantages like the reduction of radiation exposure, the usage of less invasive surgical approach, or shorter interventions due to better planning. Extending these systems with capabilities to perform biomechanical simulations the surgeon has the possibility to predict the mechanical behavior of the bones and the implants before the surgical intervention. Most of the existing systems are not designed to be used by a surgeon or do not provide mechanical simulation [5, 6].

Our goal is to develop a surgical planning tool that is used by the surgeon before the operation to plan the intervention. The tool is able to perform biomechanical tests and simulate possible surgical solutions and calculate their mechanical behavior. This information helps the surgeon to select the most suitable treatment for the given patient.

2. Method

Our method takes CT images as input, which are available in the DICOM standard format. First the segmentation is done, where the voxels are labeled depending on which bone fragment they belong to. Here, a semi-automatic segmentation algorithm is used which is based on fuzzy connectivity [1]. The algorithm is semi-automatic because the user has to set some seed points prior to segmentation. In a post-processing step, the holes and cavities are filled, to avoid eventual inner surfaces. In the second step, the Marching Cubes algorithm [2] is used to generate the surface of the segmented volume of every bone fragment. This algorithm produces good quality meshes, but the number of used triangles is very high. This is not a problem for the rendering engine, but later the mesh is used as the basis for the finite element analysis (FEA) which is very resource intensive. In the next step the mesh is simplified by a decimation algorithm [3] to reduce the number of triangles. The output is now a simplified triangle mesh in 3D, describing the surface of the bone fragments seen on the CT images (see Fig. 1). This geometrical model serves as basis for the surgical planning module.

Since the bone fragments could have moved during the fracture, it is necessary to relocate them to their original places. The surgical planning module provides a user interface to move and rotate the fragments in 3D space via the mouse. The planning module is also the place where virtual implants are inserted. The fixation screw is implemented as a parametric object with parameters like length, radius, head size, thread length, pitch, etc. The fixation plate has just width and height as parameters, but it follows the surface as the user clicks with the mouse. A typical surgical plan is presented on Fig. 2.

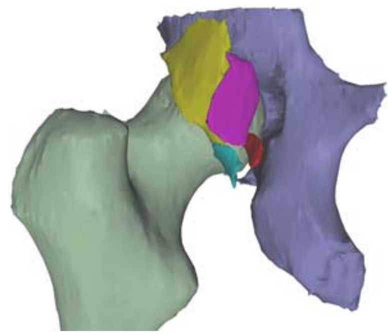

Fig. 1 The geometrical model of the femur and broken bone fragments. *Colors* are used for better distinction

To verify the mechanical behavior of a surgical plan the geometrical model has to be converted into a mechanical model. This is done by exporting the geometry as a finite element mesh, extending it with material properties, adding load and adding boundary conditions at certain areas. The finite element mesh is constructed from the triangle mesh where every triangle corresponds to a 3-node shell element. This approximates the outer 1 mm thick bone tissue, called cortical bone. This surface of the bone is 100 times stiffer than the inner cancellous bone, so the material properties are set to the values of the cortical bone. The values for the modulus of elasticity and the Poisson's ratio are taken from the literature [4]. The addition of load and boundary conditions is handled by the user interface of the planning module. To connect the different objects on the level of the finite element mesh 2-node elements are automatically inserted between nodes which are close to each other.

The finite element analysis is performed by external software, and the communication is implemented through session files. The results of the analysis, namely the deformation under the load and the material stress are presented in our system (see Fig. 3). The deformation of the material is multiplied by an appropriate high value and displayed as an animation to get a better understanding. The material stress is encoded as color information. Based on these images the surgeon can see the week points of the surgical plan, which are basically areas with high material stress. Alternatively he can compare the mechanical behavior of two or more other plans, which helps him to find the right fixation treatment for the given patient.

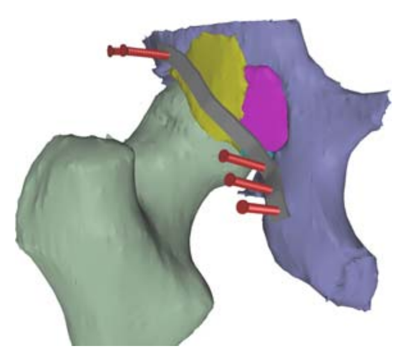

Fig. 2 Surgical plan of the hip presented at Fig. 1. The fragments are repositioned and a bending plate is used with five screws to fixate the fracture. The screws are not inserted into the bone for better visibility

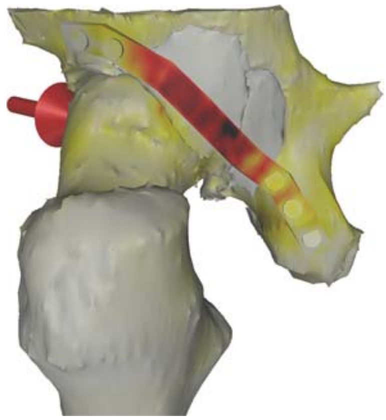

Fig. 3 The result of the finite element analysis. The loaded area and the direction of the load are indicated by the *red arrow* on the *left*. The *back side* of the largest pelvic fragment is made unmovable (zero displacement). This simulates the force which could have caused the original fracture. *White colors* indicate low material stress, and *red colors* indicate high material stress in a certain area

3. Results

With this system a virtual biomechanical laboratory was created. Various studies were carried out including pelvis, hip, knee, wrist and jaw. Three different fixation methods of a fractured pelvic bone were compared for stability. Some frequently occurring fracture patterns of the pelvic bone were simulated to get a better understanding of the fracture mechanism.

We validated our method by comparing our simulation results to previous cadaver studies on healthy pelvic bones. In this cadaver experiment the displacements of several point pairs were measured. Repeating the study virtually showed comparable results.

The software runs on a Pentium IV PC with 1 GB RAM and even the most time consuming task, the finite element analysis does not take longer than 30 s. A complete study with all user-interactions including the segmentation, the bone repositioning and the implant insertion can be performed in 20 min.

4. Conclusion

A surgical planning tool was presented which is also capable of performing biomechanical simulations. The tool can be used by the surgeon to create surgical plans for difficult fracture fixation and to predict the biomechanical stability of the planed surgery before the operation. Besides his own surgical experience, the surgeon can rely on numerical computations when selecting the right fixation type. The research of new fixation types and their comparison to existing ones is also a good application of our method. The system is also frequently in use in the post gradual education field to demonstrate the effects of high energy impacts on bones.

References

1. Nyúl LG, Falcao AX, Udupa JK (2000) Fuzzy-connected 3D image segmentation at interactive speeds. SPIE Med Imaging 3979:212–223
2. Lorensen WE, Cline HE (1987) Marching cubes: a high resolution 3D surface construction algorithm. ACM SIGGRAPH Comput Graph 21(4)
3. Schroeder WJ, Zarge JA, Lorensen WE (1992) Decimation of triangle meshes. In: Proceedings of SIGGRAPH, pp 65–70
4. Beer FB, Johntson ER (1980) Mechanics of materials. McGraw-Hill, New York, 42
5. Mimics: http://www.materialise.com/mimics/
6. SQ Pelvis: http://www.sekvenca.com/en/sekvenca-medical/

Combined statistical model of bone shape and density for orthopaedics

Laura Belenguer Querol[a], Mauricio Reyes[a], Philippe Büchler[a], Nils Reimers[b], Daniel Rueckert[c], Lutz P. Nolte[a], Miguel A. González Ballester[a]

[a]MEM Research Center, Institute for Surgical Technology and Biomechanics, University of Bern, Switzerland

[b]Stryker Osteosynthesis, Kiel, Germany

[c]Visual Information Processing, Department of Computing, Imperial College London, UK

Abstract We propose a statistical bone model for improving current orthopaedic implant design and surgical techniques. The combined statistical model of shape and density describes the properties across a given training population of CT scans. These properties allow to improve the design of implants by testing them on virtual bone instances generated from the statistical model. The model is built using principal component analysis on the CT images (in which Hounsfield units are proportional to bone density). This statistical model contains the average bone and the principal modes of variation, spanning a probability density function in a combined shape and intensity space, which allows generating further valid instances of bones.

Keywords Statistical models · Registration · Orthopaedics

1. Introduction

Current orthopaedic implant design and surgical techniques make use of limited information about the target bone. The evaluation and implementation of the implant, to find the optimal implant shape, is based on expert knowledge and a trial-and-error process, usually performed on a set of cadaver bones by manual fitting and fixation procedures. No statistical statements about the appropriateness of implants regarding their shape and mechanical properties across the patient population can be derived from this process. Thus, it is difficult to assess whether the set of implant shapes will fit most of the target population and no prior considerations about the density of the femur bone are given to predict the mechanical response to the implant. This inaccuracy could lead to implant loosening or wear, thus leading to revision surgery. In this work we propose a combined statistical model of bone shape and density (CT intensity) that allows to virtually test the implant on a representative set of bones. This will ease the design and improve the accuracy of the implant to ensure a better fitting of the implant in the target population.

2. Methods

The first step in the model construction comprises the segmentation of the original CT image data to isolate the target structure, in our case the proximal femur bone. Based on this segmentation, a mask is applied, so only voxels within the region of interest are considered for further processing. One of the training set images is taken as the reference image and the remaining instances are registered by a similarity transform (translation, rotation and scaling) to compensate for the differences in size and bone positioning during CT acquisition. The reference image can be described as:

$$C_{\mathrm{REF}} = (x_1, y_1, z_1, I_1, \ldots, x_n, y_n, z_n, I_n)^T.$$

To establish the correspondences between the shapes, the reference image is deformed non-rigidly [1] to the previous globally aligned images. Based on the deformation fields obtained from the registration process, we build vectors of corresponding positions in an analogous form to the reference vector, by adding the deformation fields in each direction to the corresponding reference coordinates. The intensity values are read in the target images at the resulting coordinates [2]. The target image is defined as: