# MedEdit: A Computer Assisted Planning and Simulation System for Orthopedic-Trauma Surgery


Krisztián Ollé*
Balázs Erdőhelyi†

Department of Image Processing and Computer Graphics,
*Department of Trauma Surgery
University of Szeged, Hungary



## Abstract

Skeletal injury operations are in general of high complexity and require extreme accuracy. That is why it seems practical that prior to a surgical intervention a geometric and mechanic model is prepared, which can be used to simulate various operational solutions. We present here a computerized system, which we call MedEdit, that helps the surgeon to plan the operation and with the use of a Finite Element Analysis (FEA) program the effects of the modifications can be measured or compared.



**Keywords:** Image processing, Surgical planning, Finite Element Analysis (FEA)


## 1 Introduction

A team of experts has been assembled from Department of Trauma Surgery and Department of Image Processing and Computer Graphics of University of Szeged. The goal of the team is to research and develop appropriate software and procedures capable of performing biomechanical tests and diagnosis on newly injured (human) accident victims with bone damage, without surgical intervention. Presently the trauma-orthopedic surgeons can use only X-ray and CT images and their own clinical expertise to evaluate the type of surgery required to stabilize the bone (also dependent on amount, type and composition of the material). Mechanical modeling would provide accurate data that reflects the stability of the osteosynthesis of the patient before surgical procedure. Several complications could be avoided by using this type of biomechanical computer modeling, while a more accurate and immediate assessment capabilities could be provided for surgeons. This method will provide new possibilities that would complement current visual analysis methods and it will also provide meaningful possibilities in the postgraduate education field. Using these methods new innovations connected to the bone surgery can be tested using computer procedures, thereby reducing the required human biomechanical experimental testing; eliminating associated high costs and time loss.

There are systems for solving such or similar surgery problems, see for example [7]. They are using 2D or 3D imaging devices to collect the necessary data for treatment planning. Our system solves this problem using CT images and we create a geometric and then a mechanical model for studying the possible surgery solutions. The mechanical tests are performed by a Finite Element Analysis (FEA) software. In this paper the system and the implemented steps are presented.

## 2 Description of the System

To solve the mentioned problems a system is needed to support the surgeon at the stage of surgical planning.

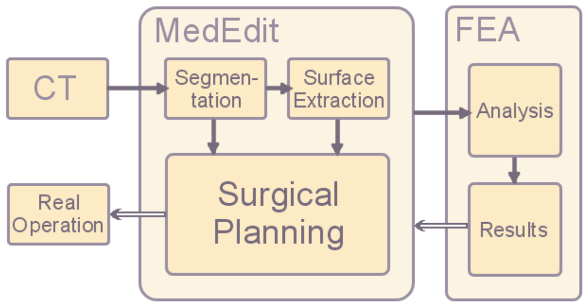


Figure 1: The schema of the system with the main compenents and the connections between them. Filled and empty arrows represent data flow and visual feed-back, respectively

Figure 1 shows the main components of the system (see also [4]). Computer Tomography (CT) images are available in DICOM (Digital Imaging and Communications in Medicine) format. Our MedEdit software reads these images and stores them in memory. The segmentation is based on the grayscale values of the image representing the absorption of the tissues. Since the density of bones


*ollek@inf.u-szeged.hu
†ber@inf.u-szeged.hu

differs from other tissues (such as blood or muscle), usually it is possible to define a threshold-window to segment bone tissue.

Next the surface extraction follows. Here we use the marching cubes algorithm [8], which creates triangle models from 3D segmented data. This algorithm produces high number of nodes and triangles, so we continue with a surface simplification algorithm [6] to reduce our model. We also keep the segmented 3D volume for further processing.

After these steps, the surgeon can perform a virtual operation on the model by joining broken bone parts, drilling cylindrical holes and inserting screws or implants into the holes drilled in bones.

By adding material properties to our geometric model, a mechanical model is created, which can be used to perform a Finite Element Analysis (FEA). In order to make stress studies load and displacement vectors and boundary conditions are given to the mechanical model. Usually FEA requires special engineering knowledge. In our system the surgeon can use it easily, without this expertise, via a session script interface between MedEdit and FEA.

The stress analysis is done in the FEA software and the results are also presented there, or can be loaded back into MedEdit. Depending on the results the surgeon can verify his strategy, or work out a new plan starting the procedure again from the virtual operation. It is possible to test different options and analyze the consequences virtually.

## 2.1 MedEdit System

The system builds up from modules (see Figure 2). Each of them are specialized to some special task. The first module imports the DICOM images and segments the bone from the gray scale CT scans. Then the 3D structure is constructed from the segmented volume model. Usually we get a very complex geometrical model, so we use a mesh simplification algorithm to eliminate the complexity of the surface. In the fourth module we created a medical surgery planner, where the surgeon can test several surgical solutions. We implemented different kinds of 3D editing functions like implant insertion, drilling, and slicing. Then the surgeon can apply forces to the model in the exporter module and export the data to the FEA system.

## 2.2 Segmentation

In this module the aim is to separate the bone tissue from the other tissues. This means that we have a 3D grayscale volume and we create a 3D binary volume, where the bone and background voxels are represented by 1 and 0, respectively.

First we tried to use different thresholds for the bone segmentation. This method is useful when the bone can be easily separated from the muscles. But in the case of the pelvis this method did not work, because other tissues surround the pelvis having similar color as the bone. So we use another segmentation method, which does not need too much user interaction. Finally, we found that fuzzy connected 3D image segmentation algorithm [5] is good for our needs (see Figure 3). The algorithm needs some seed points in the bone tissue then it segments the whole volume having similar property as the neighborhood of the selected points. After the segmentation post-processing is needed to fill the cavities on the slices, because the eventual inner surfaces are not interesting in further steps.

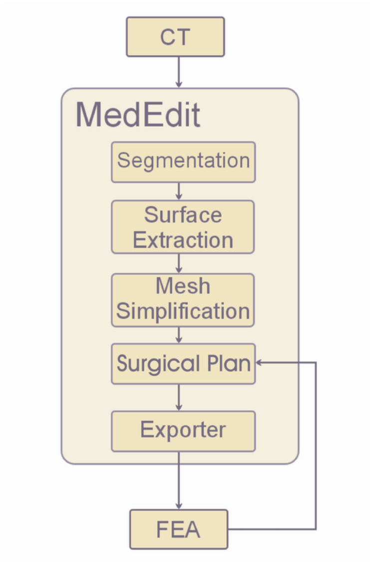


Figure 2: The schema of the MedEdit system with the main components and the connections between them

Figure 3: The steps of the segmentation. A CT slice with a part of the pelvis (left). The points selected by the fuzzy segmentation algorithm (middle). The result of the segmentation after cavity filling(right)

## 2.3 Surface Extraction

For the surface extraction we tried to use contour extraction with slide-by-slide contour simplification followed by a triangularization [1]. But we found that the marching cubes algorithm [8] fits better to our needs and it is more robust. The only drawback of this algorithm is that it produces a high number of triangles (see later Figure 8).

## 2.4 Mesh Simplification

We need mesh simplification for several reasons. Besides performance improvements for the rendering engine we have a strict upper limit for the number of triangles accepted by our FEA software. To generate the reduced mesh we use the surface simplification algorithm by Garland et al. [6] (as an example, see Figure 9).

### 2.5 Surgical Planning

In this module the system provides a surgical planner interface for the surgeon. He can plan and simulate several surgical procedures. He can also check the FEA result here. It is possible to assembly the broken bone parts by dragging and moving them with the mouse. We are using collision detection to simulate a real life behaviour of the bones (Figure 10).

When all bones are on their positions, virtual implants can be used to fix them together. For example, a hole can be drilled into the bone, then a screw-implant can be inserted into the model. For the exact positioning the user interface gives many ways to select the most appropriate view, angle, and magnification for the presentation of the 3D object (Figure 11). In order to visualize the very complex structures, the surgeon has the possibility even to define transparent bone properties (similar to the X-ray transparent bone tissue).

## 3 Communication between MedEdit and FEA

The geometric model can be exported into the FEA system. To complete the geometric model to a mechanical one, several material properties, as load and boundary conditions should be added to the finite element mesh (Figure 12). Our MedEdit software provides an easy-to-use way defining all these properties.

At this point we have two data sources: The surface geometry and the segmented 3D volume. The earlier is treated by the FEA system as a shell, the latter as a solid body. MedEdit can export both of them.

### 3.1 Shell model

When exporting the surface geometry, 3-node shell elements are inserted into the finite element mesh for every surface triangle (Figure 12). This mesh is the base for the computations of forces, displacements and deformations. The shell elements are triangular thin shells with bending capabilities and constant thickness.

The shell model simulates the outhermost 1 mm thick part of human bones, namely the cortical bone wich is 100 times stiffer than the inner cancelous bone. The material type of the shell elements is set to the property of the cortical bone: modulus of elasticity E=1100 MPa and Poisson's ratio of 0.3 is used. These values are based on cadaver studies [2] and the literature [3].

Unfortunately adding implants to the analysis is difficult, because the model would allow unwanted movements. For example a drill had contact only at the surface and the tip could move in any direction.

### 3.2 Solid Model

When exporting the segmented 3D volume, 8-node solid elements (cube-like elements) are inserted into the mesh for each bone-labeld voxel. We use a simple sub-sampling algorithm to reduce the number of voxels keeping the computational resources (time and memory) acceptable.

Implants, wich are placed into the model during the virtual operation are recalculated, and inserted into the 3D volume. This way, implants will be added to the finite element mesh too, see Figure 4.

In this case the material properties are based on an average bone material i.e. modulus of elasticity E=300 MPa and Poisson's ratio of 0.2 is used.

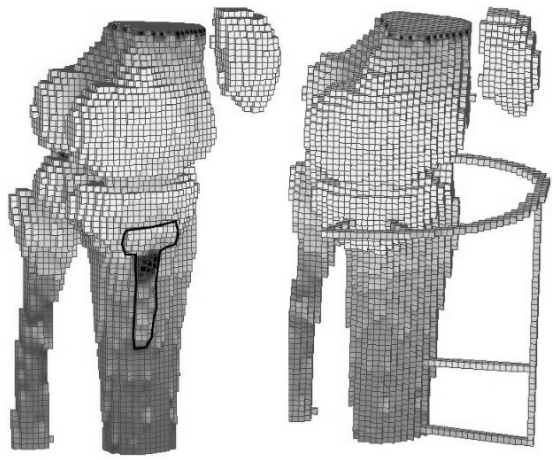

Figure 4: Results of the finite element analysis of two knie-joints with different implants. First a 8-node hexahedral mesh was created as described in Section 3.2. Next the broken bonepart was fixed with T-shaped plate (left) and a Hybrid Ring Fixator (right) implants. Dark colors represent higher material tension.

### 3.3 Mixed model

Since the solid model requires too much computational resources, we developed a mixed model, which joins the positive features of the previous two models: small in size, has inside elements, and differentiates between different bone types.

Using the segmented 3D volume and the marching cubes algorithm, a triangle mesh is created. Every such triangle corresponds to a 3-node shell element in the finite element mesh. Next, 2-node finite elements are added to the mesh to connect the nodes inside the model. From every node a ray is cast paralel to the three co-ordinate axes and a matching node is searched at the intersection on the other side of the model (Figure 5). If a 2-node element would be to long, interior points are added.

The 2-node elements simulate the inner bone structure preventing the implants from undesired movements.

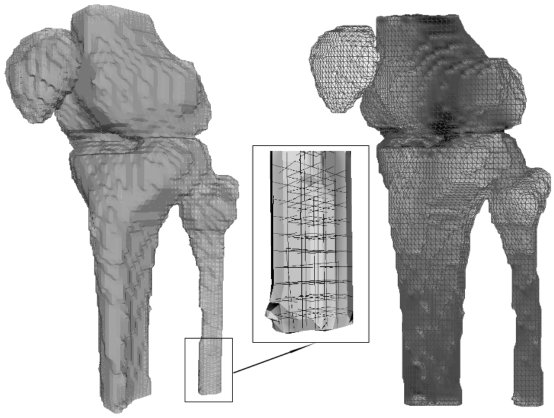

Figure 5: On the left the mechanical model of the knie-joint. The finite element mesh was generated as described in section 3.3. The outer 3-node elements are made transparent, to be able to see the inner 2-node elements. On the right, the result of the analysis

The outer elements represent the cortical bone and their material properties are set accordingly. The material constants of the cancelous bone is used for the inner elements (E= 10 MPa, Poisson's ratio 0.3).

# 4 Results

The system works presently in an experimental way. It is able to perform all tasks, but there are still points where some user interaction is needed. For example, the segmentation starts by setting seed points manually, its result should be checked by the surgeon. The communication with FEA is not automatic; it is solved by a session file.

The system is implemented and works. Generally, it is able to create the geometric and mechanical models in ca. 5 minutes including the user interactions. The FEA takes roughly 6 minutes for a 3D pelvis volume study (on a 2 GHz computer with 1,5 GB memory).

Our stress results seem to match the clinical expectations, although quantitative tests and measurements are still to be done.

## 4.1 Example

A complete example is presented using the generated images. The steps of the procedure can be followed through the Figures 6-13. The user interface is not visible here.

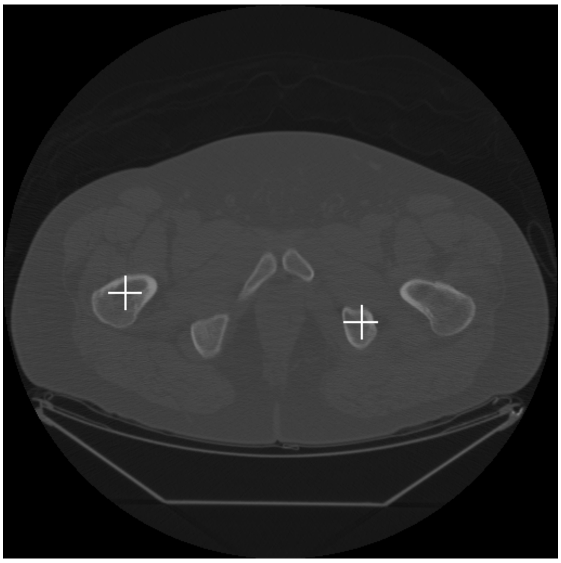

Figure 6: The user indicates with crosses the positions of the seed points for segmentation

Figure 7: The result of the segmentation of the bone in Figure 6

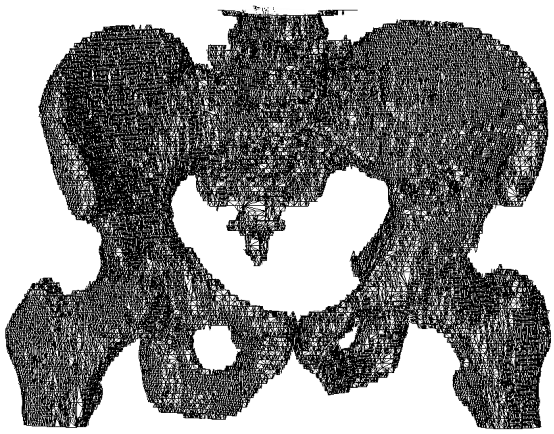

Figure 8: The pelvis produced by the marching cubes algorithm consisting of more than 200.000 triangles

# 5 Future Plans

## 5.1 Navigation

We plan to extend our system with the ability to help the surgeon during the operation to find the right points and

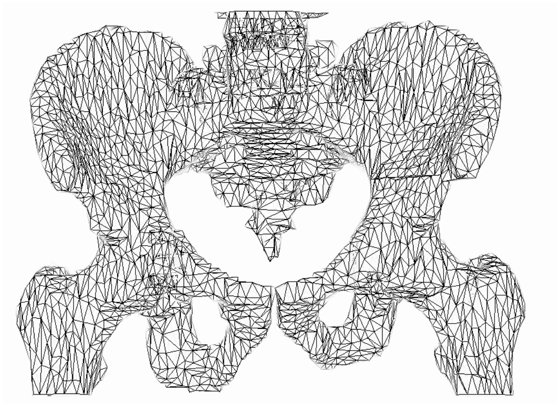

Figure 9: The same pelvis as in Figure 8 after surface simplification. The surface consists of more than 10.000 triangles

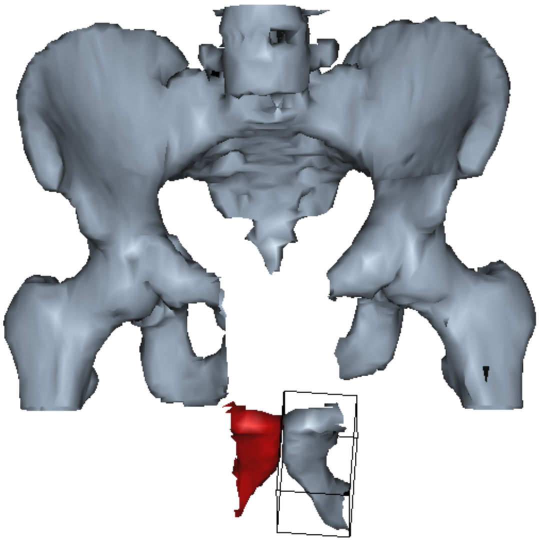

Figure 10: Assembling the simulated boneparts with collision detection. The selected bonepart is surrounded by its bounding box and can be moved virtually. If a collision occures the other bone is highlighted

angles. With three or more cameras installed in the operating theatre we could identify some special marked points and give real-time information where and in which angle the surgeon has to insert the implants.

## 5.2 Education

As we mentioned above, the orthopedic-trauma surgeon can generally use only X-ray, CT, MRI images and their own clinical expertise to estimate the type of required implants to stabilize the bone for the appropriate biomechanical stabilization. Mechanical modeling is able to provide expidite data that reflects the stability of the osteosynthesis

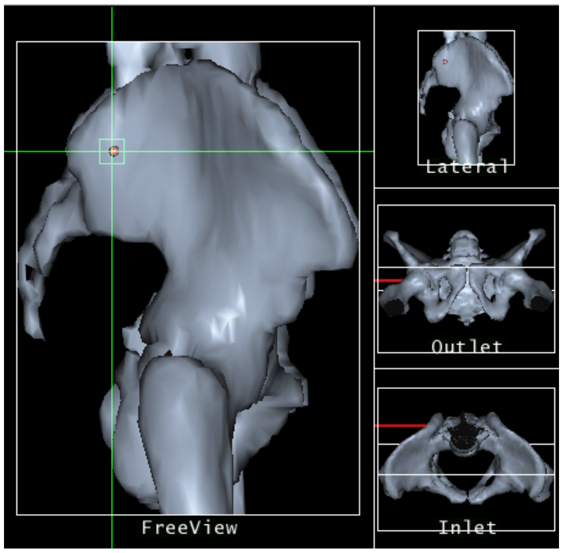


Figure 11: Orientation images for surgical planning. Lateral view of the pelvis (left) with a screw positioning tool and 3 other views of the same object (right)

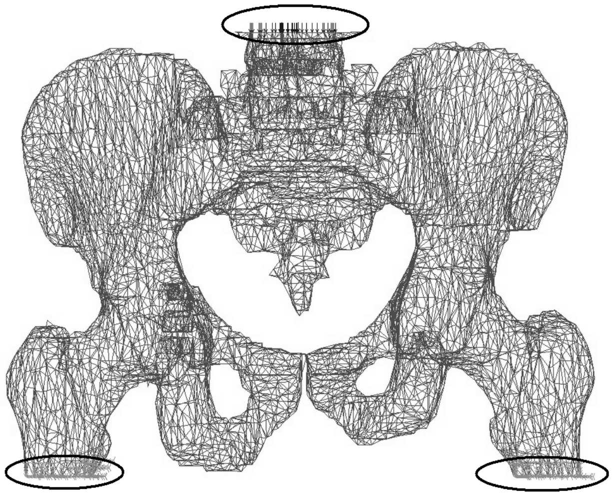

Figure 12: The mechanical model of the surface of the pelvis generated from the geometric model just before the stress analysis. It consists of a mesh, applied forces, so-called zero displacements (positions without displacements), and material properties. In this case the FEA mesh consists of so-called 3-node shell elements (triangles). The positions of the applied forces are indicated by an ellipse on the top. The positions of the zero displacements are indicated by two ellipses on the femurs

of the patient before surgical procedure. Several complications could be kept off by using the above-mentioned type of biomechanical computer modeling, while a more scrupulous and prompt assessment capabilities could be given for surgeons. This method will provide new pos-

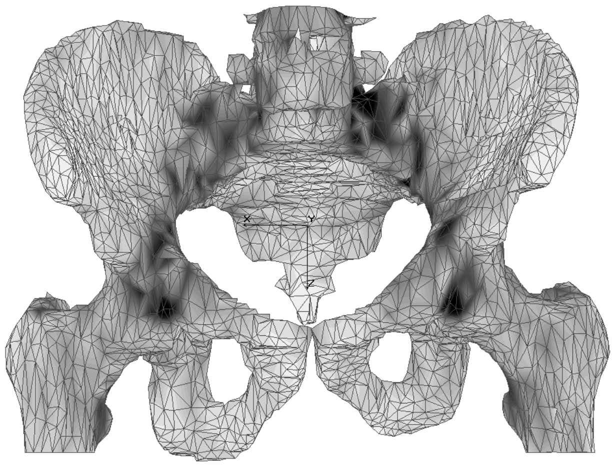

Figure 13: The result of the stress analysis on the object of Figure 12. Dark colors show areas with high material stress

sibilities that would complement current visual analysis methods and it will also do for meaningful perspective in the postgraduate education field.

## Acknowledgement

This work was supported by the Hungarian grant OTKA T37840. The authors wishes to thank Endre Varga and Attila Kuba head of department and project leader.

## References

[1] Geiger B. *Three-dimensional modeling of human organs and its application to diagnosis and surgical planning*. INRIA, 1993.

[2] Varga E. *Biomechanical and Clinical Investigation of Pelvic Ring Injuries*. Annals of the Albert Szent-Györgyi Medical and Pharmaceutical Center, 2000. vol. 47.

[3] Beer FB and Johntson ER. *Mechanics of Materials*. McGraw-Hill Inc., 1980. 42.

[4] Endre Varga Csongor Halmai Attila Kuba Krisztián Ollé, Balázs Erdőhelyi. *MedEdit, a Computer Assisted Planning System for Orthopedic-Trauma Surgery*. Proceedings of the 25th International Conference on Information Technology Interfaces., 2003. p. 507-512.

[5] Nyúl LG, Falcao AX, and Udupa JK. *Fuzzy-Connected 3D Image Segmentation at Interactive Speeds*. SPIE Medical Imaging, 2000. 3979 p.212-23.

[6] Garland M and Heckbert PS. *Surface simplification using quadric error metrics*. ACM SIGGRAPH' Computer Graphics, 1997. p. 209-216.

[7] Robb RA, Hanson DP, and Camp JJ. *Computer-aided surgery planning and rehearsal at Mayo Clinic*. Computer, 1996. 29(1):39-47.

[8] Lorensen WE and Cline HE. *A High Resolution 3D Surface Construction Algorithm*. ACM SIGGRAPH Computer Graphics, 1987. 21, (4).