

MedEdit: A Computer Assisted Image Processing and Navigation System for Orthopedic Trauma Surgery*

Krisztián Ollé[†], Balázs Erdőhelyi[†],
Attila Kuba[‡], Csongor Halmai[‡] and Endre Varga[‡]

Abstract

The surgery of fractured bones is often a very complex problem. That is the reason why it would be beneficial to create a geometric and mechanic model of the bones before surgical intervention. The model geometry is based on the CT images of the patient and the known physical properties of the bone. A computerised system is presented here, called MedEdit, which helps a surgeon plan an operation. The system includes a Finite Element Analysis (FEA) program to measure the stress effects of the possible surgical solutions. Following the simulation and analysis of the behaviour of the modelled bone, surgeons can find the best surgical solution for the patient.

1 Introduction

The surgical repair of fractured bones is often a difficult task, and the fixation of these bones has to be planned very carefully. This is why trauma surgeons may use Interactive Image-Guided Surgery (IIGS) or Computer Aided Surgery (CAS) systems to improve surgical accuracy. By using these systems treatment can have several advantages. For example one can have a less invasive surgical approach, have less time consuming interventions due to better planning, and there can be a reduction in radiation exposure.

CAS technology was first introduced in neurosurgery. Here the neurosurgeons focused on pre-operative planning and an increase in intra-operative accuracy. By applying external frames to the patient's skull there is a possibility of locating targets within the brain to a high precision. This technique is called stereotactic neurosurgery [7]. The first application of CAS in orthopedic and trauma surgery

*This work was supported by the Grant OTKA T37840

[†]Department of Image Processing and Computer Graphics, University of Szeged, H-6720 Szeged Árpád tér 2. E-mail: {ollék, ber, kuba, halmai}@inf.u-szeged.hu

[‡]Department of Trauma Surgery, University of Szeged, H-6720 Szeged, Semmelweis u. 6. E-mail: varga@trauma.szote.u-szeged.hu

was in 1995 for the placement of lumbar pedicle screws [8]. After, this area started to develop rapidly and nowadays there are commercial systems available for solving such surgery problems (see [5], for example). There is a system [9] which focuses on image segmentation, but it has a finite element analysis (FEA) option and navigation modules.

Most of the existing systems are not capable of performing mechanical simulations or are not designed to be used by a surgeon. They have complicated user interfaces meant for an experienced computer user, but surgeons are not usually that experienced. A more user-friendly user interface is required.

Our goal here is to develop a suitable CAS package that is capable of performing biomechanical tests almost in real time and help in the diagnosis of patients who have sustained bone damage. The program can simulate possible surgical solutions and calculate their results. It can also present different stress effects.

This paper describes our ongoing work on computer aided surgical planning and developing a tool for it. In [2] we presented an earlier version of our system, which could perform virtual editing and analysis but had limited capabilities. Our new version has more options and is more user-friendly.

2 Description of the System

The parts of our MedEdit system intended to help the surgeon at the stage of surgical planning are shown in Fig. 1.

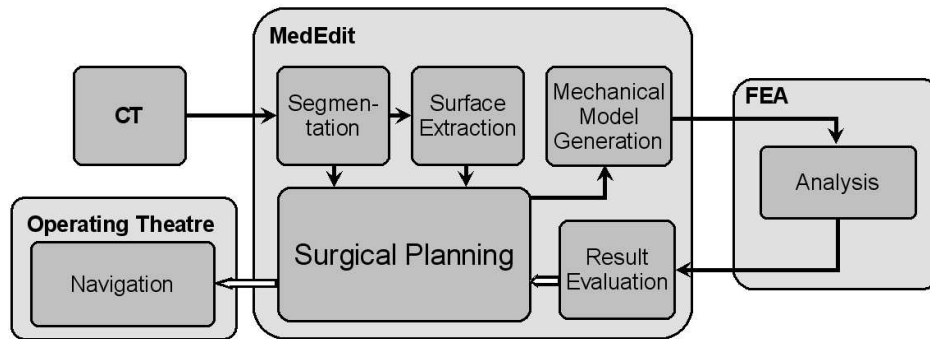


Figure 1: The system schema with its key components and connections. Filled arrows and empty arrows represent data flow and visual feed-back, respectively

Three light grey boxes can be seen here which are the main components of the system. In the MedEdit box there are smaller boxes which we will call modules. For input Computer Tomography (CT) images are available in DICOM (Digital Imaging and Communications in Medicine) format - a common medical imaging standard. MedEdit first reads these images and then stores them in the memory. In

order to generate a mechanical model of the bones, the CT images are segmented, i.e. the pixels representing the bone tissue are separated from the others.

After segmentation information about the object is stored in two data structure formats. The first represents segmented image data as a volume, while the second represents the surface of the data as a triangular mesh. Then the surgeon can perform a virtual operation on the model (the virtual object) by joining broken bone parts, drilling cylindrical holes and inserting screws or implants into the virtually drilled holes with the Surgical Planner module.

By adding material properties to our raw/geometric model in the mechanical model generator a mechanical model is created which can be used to perform a Finite Element Analysis (FEA). Usually using an FEA program requires special engineering knowledge of the material properties and it often has a complex command line user interface. Hence it is really hard to configure and work with. With our system the surgeon can use an FEA program without needing any special expertise. This is because we designed a graphical, mouse-driven user interface in cooperation with surgeons from the local Traumatology Department.

The stress analysis is done using a FEA program and the results are presented in MedEdit. Depending on the results, the surgeon can verify his strategy, or work out a new plan by starting the procedure again from the virtual surgical operation step. It is possible to test different options and analyse the consequences in a virtual environment. Currently we are working on a surgery navigation component, which is in the experimental phase. This could be of use to a surgeon inside the operating theatre after he has planned the operation.

3 Surgical planning in MedEdit

The MedEdit program has a simple modular design, each of the modules performing a special task. The first module imports the DICOM images and segments the bone from the grey scale CT scans. Then a 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 reduce the complexity of the surface. In the fourth module we created a medical surgery planner where the surgeon can try out several possible surgical solutions before performing the actual operation. We implemented various kinds of 3D editing functions like implant insertion, drilling, and slicing. The surgeon can apply forces to the model in the mechanical model generator and then export the data to the FEA system.

3.1 Segmentation

CT scans represent data as greyscale values, these values depending on the absorption of the tissues. Since the bone tissue density differs significantly from other materials like blood and muscle, it is usually possible to separate them. This means that we have a 3D greyscale volume and we create a 3D binary volume where the bone and background voxels are represented by binary-valued data (ones and zeros).

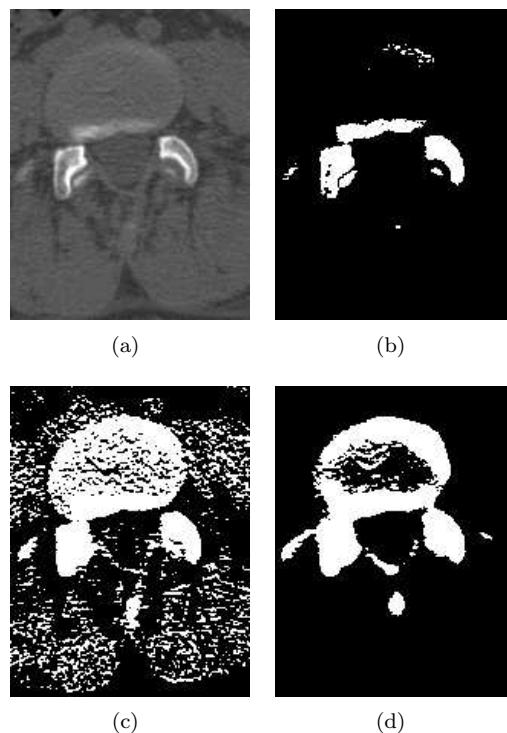


Figure 2: A typical situation when fuzzy segmentation achieves better results than global thresholding. Fig. (a) is the original image of the spine, while Fig. (b) shows the same image with a high threshold value. In this case the bone part is only partially visible. Fig. (c) shows the results with a low threshold value, with a lot of noise appearing around the bone. Fig. (d) shows the same image, but it was segmented by using a fuzzy algorithm.

At first we tried out different thresholds for the bone segmentation (see Figs. 2(b) and 2(c)), but then our experiences showed that this method was only useful when the bone could be easily separated from the muscles. However, in the case of the pelvis this method did not work because other tissues surrounding the pelvis have similar grey values to bone tissue. So we looked for another segmentation method that is just as easy for the user as the thresholding algorithm is. Later we found that a fuzzy connected 3D image segmentation algorithm [3] is good for our needs and can achieve good results (see Fig. 2(d)). The algorithm requires some seed points in the bone tissue then it segments the whole volume containing points having similar properties as the neighbourhood of the selected points. After the segmentation part post-processing is needed to fill in the cavities present in the slices, because inner surfaces are not of interest to us in later steps (see Fig. 3).

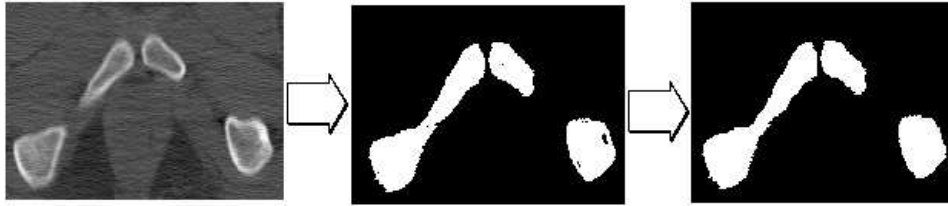


Figure 3: The steps of the segmentation phase. A CT slice showing part of a pelvis (left); the points selected by the fuzzy segmentation algorithm (middle); the results of the post-processing phase (right) after filling in the cavities

3.2 Surface Extraction

For surface extraction and geometry building we tried using contour extraction with slide-by-slide contour simplification followed by a triangularisation [1]. This method works well on tubular shaped objects like a trachea, but fails on complex objects like a human pelvis and requires a lot of effort to convert the data to a triangular format. Later we found that the marching cubes algorithm [6] was ideal for our needs and is more robust. The only drawback of this algorithm is that it invariably produces a high number of triangles (see Table 1).

Table 1: Triangular surfaces generated by the marching cubes algorithm using 512x512 CT scans of different parts of the human body.

Model	Num. of CT slices	Num. of vertices	Num. of faces
Part of a hand	124	219,223	437,256
Part of a knee	109	334,802	668,606
The whole pelvis	91	361,436	721,786

After generating the surface mesh we had to simplify it for several reasons. One was to improve the performance of the rendering engine, and another was because there is a strict upper limit on the number of triangles accepted by our FEA program. To generate a reduced mesh we employed the surface simplification algorithm written by Garland et al. [4] (for an example, see Fig. 4).

3.3 Surgical Planning

In this module the system provides a surgical planner interface for the surgeon. He can plan and simulate several surgical procedures and can check the FEA results as well. It is possible to assemble the broken bone parts by dragging and moving them with the mouse. We use collision detection to simulate the real life behaviour of bones (Fig. 5).

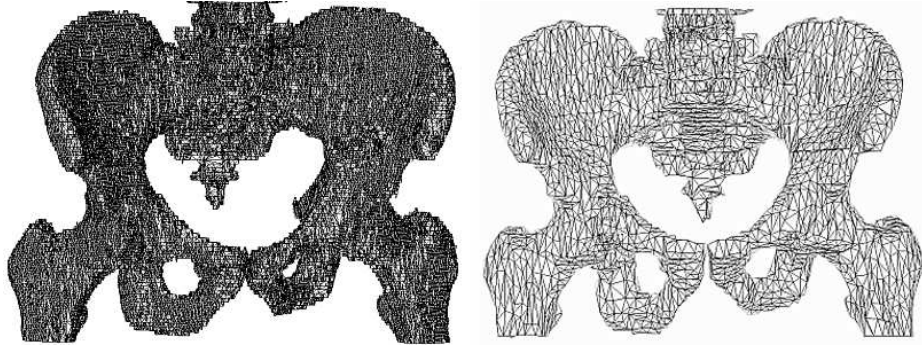


Figure 4: Geometric model and simplification of the pelvis. On the left the model consists of more than 700,000 triangles. On the right the same pelvis after surface simplification (about 10,000 triangles).

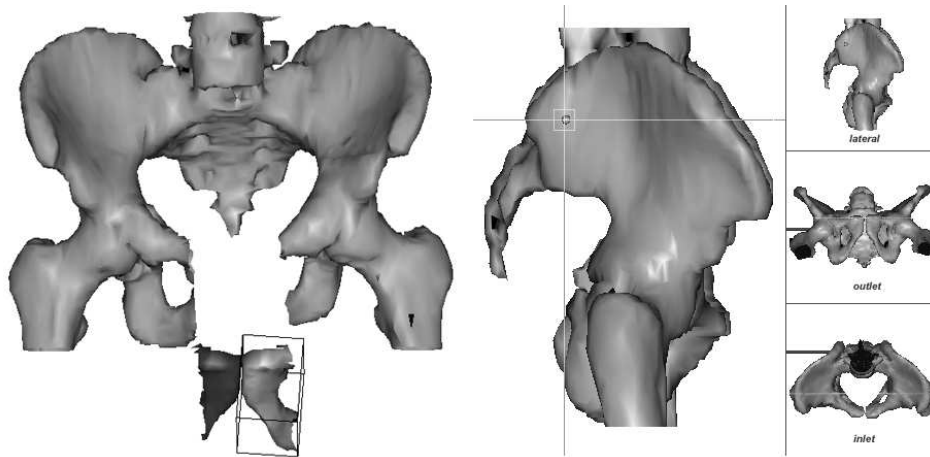


Figure 5: Assembling the simulated bone parts using collision detection (left). The selected bone part is surrounded by its bounding box and can be moved around in the virtual environment. If a collision occurs the other bone is highlighted. On the right, surgical planning using a screw positioning tool and orientation images.

When all the bones are in their positions, virtual implants can be used to hold 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 provides various ways of selecting the most appropriate view, angle, and magnification for the presentation of the 3D object (Fig. 5). In order to visualise the complex structures, the surgeon even has the possibility of defining transparent bone properties so that it will look like X-ray transparent bone tissue.

3.4 Mechanical Model Generation

To be able to export the surgical plan to the FEA system, the geometric model has to be extended to include information about material properties, load and boundary conditions. The geometric information is turned into a finite element mesh, and material properties are assigned to the finite elements which form the mesh. Nodal load vectors are set by the user to indicate load effects for certain areas of the mesh and they are then exported to the FEA system in an appropriate file format. Boundary conditions are set for the nodes, which can be fixed or move only in certain directions during the analysis. Our MedEdit system provides a user-friendly interface for defining all these properties.

4 Preparing Surgical Operations for Finite Element Analysis

For the generation of the finite element mesh we have two input data sources: the surface geometry and the segmented 3D volume. The former is treated by the FEA system as a shell, the latter as a solid body. MedEdit can export both of them for the analysis phase.

4.1 Shell Model

When exporting the surface geometry, so-called 3-node shell elements (see Fig. 6) are inserted into the finite element mesh for each surface triangle. This mesh provides the basis for the computations of forces, displacements and deformations. The shell elements are triangular thin planes with blending capabilities and constant thickness.

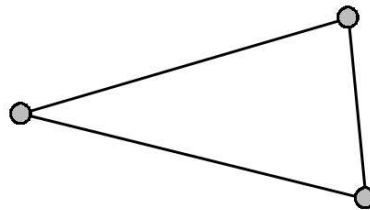


Figure 6: 3-node shell element.

The shell model simulates the outermost 1 mm thick layer of human bones, namely the cortical bone, which is 100 times stiffer than the inner cancellous bone. The material type of the shell elements is given the same physical properties as the cortical bone. Its modulus of elasticity E is 1100 MPa and Poisson's ratio is 0.3. These values are based on cadaver studies [14] and medical literature [15].

Unfortunately adding implants to the virtual model is difficult because it would allow unwanted movements. For example, a drill makes contact only at the surface and the tip could move in undesired directions. We should make it move as it does in real life.

The shell model was used to simulate some typical fractures of the pelvis to get a better understanding of the fracture mechanism. In the next two sections we will briefly describe the way the system handles some of the frequently occurring fracture patterns.

4.1.1 Lateral Compression Fracture

The lateral compression fracture of the pelvic girdle is one of the most common fracture patterns. It can occur when a big force acts on the side of the pelvis. This can happen when a motorcycle rider is hit by a car from the side at a road junction. A typical X-ray image of such an injury is depicted in Fig. 7(a), where the white arrows show the actual fracture.

Our mechanical model of this case is illustrated in Fig. 7(b). The right side of the pelvis is fixed and a force is applied from the patient's left side. In Fig. 7(c) the results of the analysis are shown.

The magnitude of the computed deformation is usually very small. Hence the nodal displacement vectors are multiplied by a suitable scaling factor to visualise the deformation. The details of the displacements are blown up so as to be clearly visible.

4.1.2 Vertical Shear Fracture

Vertical shear injuries typically occur as a result of a fall from a height, but they can also occur in motor vehicle collisions. This force puts an asymmetric axial load on the pelvis, which may result in rotational and vertical instability.

In our simulation the so-called LIV.-LV. vertebrae were fixed and a force was applied to the left femur, as shown in Fig. 8(b). The results of the analysis, which are given in Fig. 8(c), predict the same areas for a fracture as in a typical X-ray image and it also matches clinical expectations.

4.2 Solid Model

When exporting the segmented 3D volume, 8-node solid elements (cube-like elements: see Fig. 9) are inserted into the mesh for each bone-labeled voxel. We use a simple sub-sampling algorithm to reduce the number of voxels and hence keep the computational resources (time and memory) to an acceptable level. Even in the reduced case, a model with about 100,000 elements takes approximately 30 minutes to analyse. This is considerably slower than the shell model where the analysis is completed in a few seconds.

Implants which are placed into the geometrical model during a virtual operation are recalculated and inserted into the 3D volume. This way, implants will be added to the finite element mesh as well (see Fig. 10).

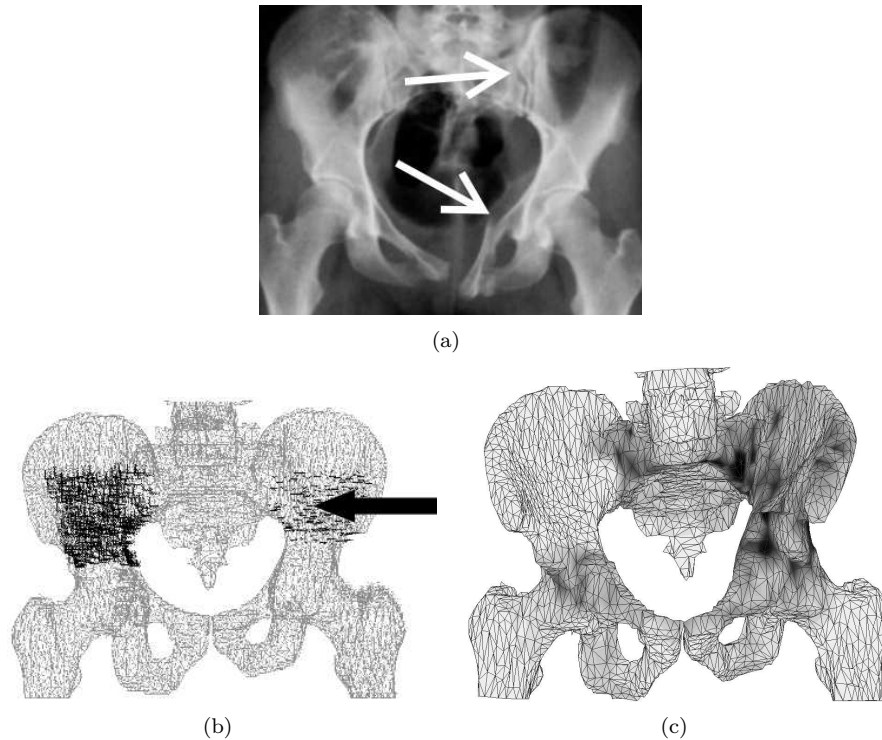


Figure 7: Simulation of the lateral compression of the pelvis. A typical X-ray image of a lateral compression fracture is shown in Fig. (a) above. Fig. (b) shows the mechanical model of the pelvis, the dark area on the patient's right side indicating the points which are fixed, while the arrow on the patient's left shows the direction of the applied force. In Fig. (c) the results of the analysis are presented. Dark areas indicate high material stress which are precisely the areas where a fracture may occur.

Here the material properties of the model are given average bone values i.e. a modulus of elasticity E of 300 MPa and Poisson's ratio of 0.2.

4.2.1 Knee study

Several tests were performed to compare two different knee fixation techniques. The knee joint of a healthy human was scanned and then imported into our system. With the editing capabilities of MedEdit one artificial fracture was created by deleting bone-labeled voxels from the segmented volume. For the fixation of the simulated fracture two different methods were investigated: internal T-shaped plating fixation, and hybrid external fixation.

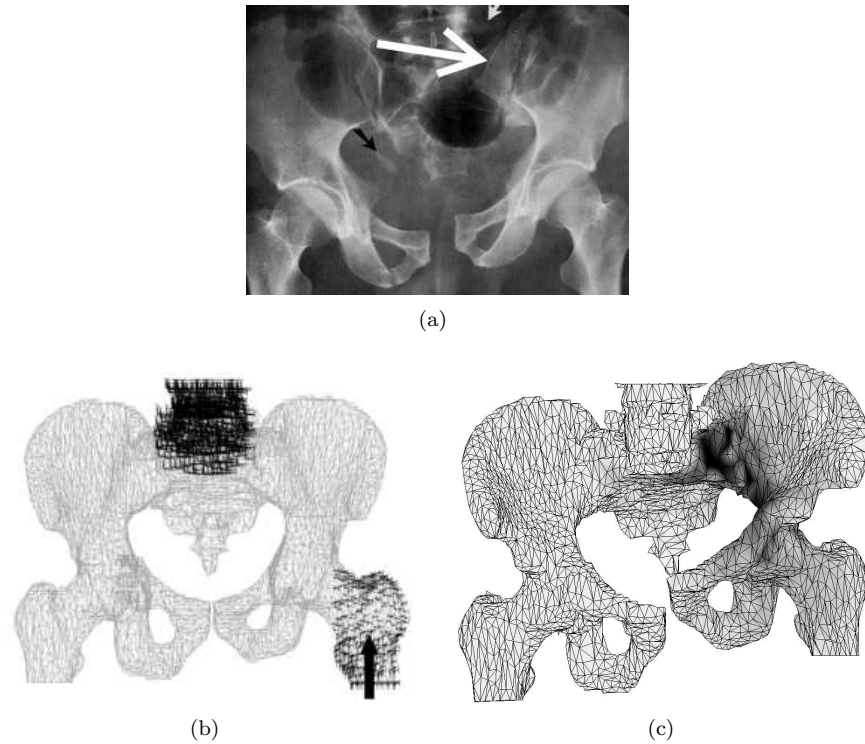


Figure 8: Simulation of a vertical shear of the pelvis. Fig. (a) shows a typical X-ray image of a vertical shear fracture. Fig. (b) depicts the mechanical model of the pelvis. The dark area in the spine region indicates the points which are fixed and the arrow on the patient's left shows the direction of the applied force. The results of the analysis are presented in Fig (c).

The mechanical model was created using 8-node hexahedral elements, a load was applied on the top nodes of the knee, and the bottommost nodes were fixed. The results of the analysis (see Fig. 10) confirm that for the internal plate fixation the implant will have high material stress near the line of the fracture. In the case of the other implant, the load is distributed along the whole fixation ring, reducing the risk of a fatigue fracture in the implant material.

4.2.2 Wrist study

A similar study as above was performed on data obtained from a scanned healthy human wrist. One artificial fracture was created on the radius, then both fixation methods were applied in the same way as we did in the knee study. The results obtained (see Fig. 11) have a similar pattern as those obtained in the knee study.

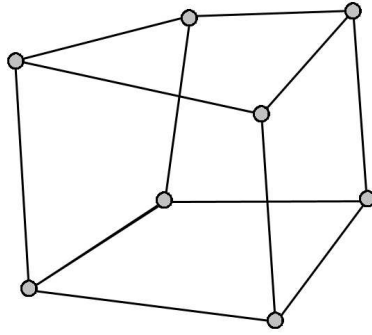


Figure 9: 8-node solid element.

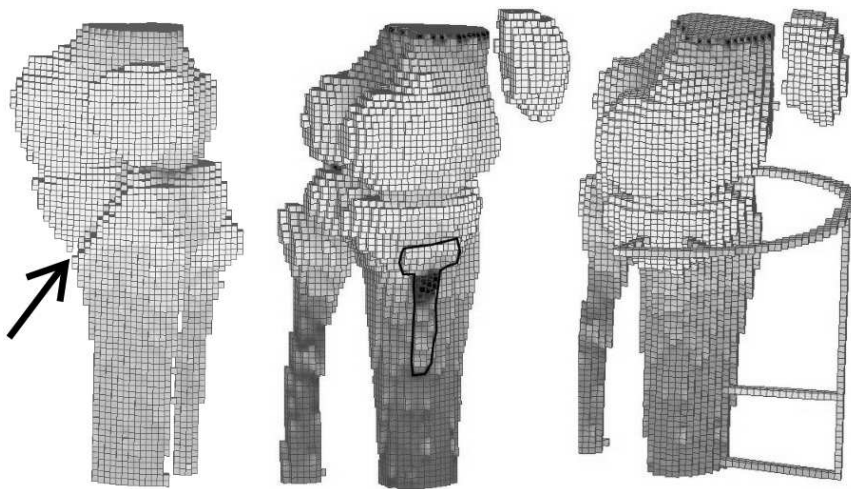


Figure 10: The results of a knee study with two different fixation types. Left, the front view of a knee joint with a simulated fracture (indicated by the black arrow). In the middle, the same knee joint viewed from the right with an interior T-shaped fixation plate. On the right, an external fixation ring is used to stabilise the broken bone.

4.3 Mixed model

Since the solid model requires a lot of computational resources, we developed a mixed model that combines the positive features of the previous two models: one that is small in size, has inner elements, and differentiates between different bone types.

First, using the segmented 3D volume representation and the marching cubes

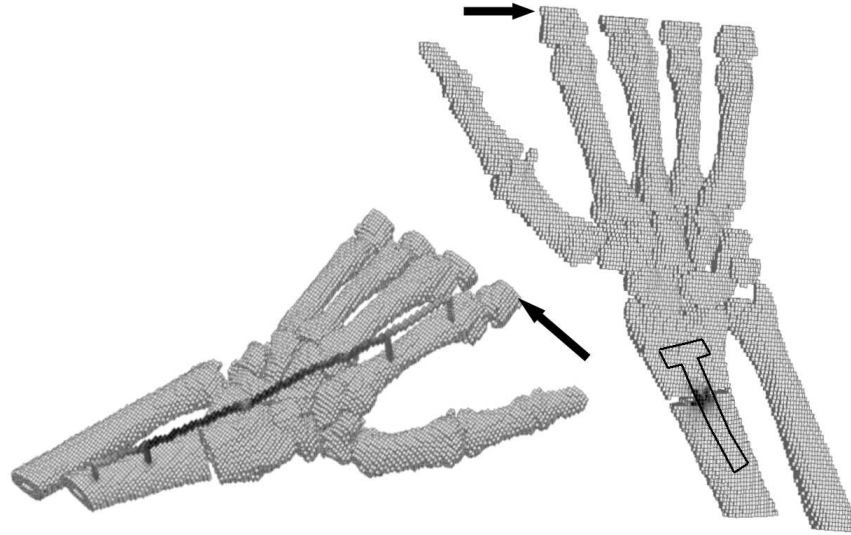


Figure 11: Results of an analysis of a wrist with an artificial fracture after applying an external fixation (left), and applying a T-shaped plate fixation (right).

algorithm, a triangular mesh is created. Each 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 each node rays are cast parallel to the three co-ordinate axes and a matching node is looked for at the intersection on the other side of the model. If a 2-node element is too long, interior points are added. The 2-node elements behave like springs and simulate the inner bone structure, thus preventing the implants from moving in an undesired way.

To see how well it works, we applied the mixed model to a knee study. The results are shown in Fig. 12.

5 Navigation

We are extending our system to help the surgeon find the right points and angles for an implant insertion during an operation. For this reason we started developing a navigation module which is capable of working with three cameras installed in the operating theatre. With this hardware we could use it to identify some special marked points and provide real-time information about where and at which angle the surgeon should insert the implants. This part of the system is now being developed by our team.

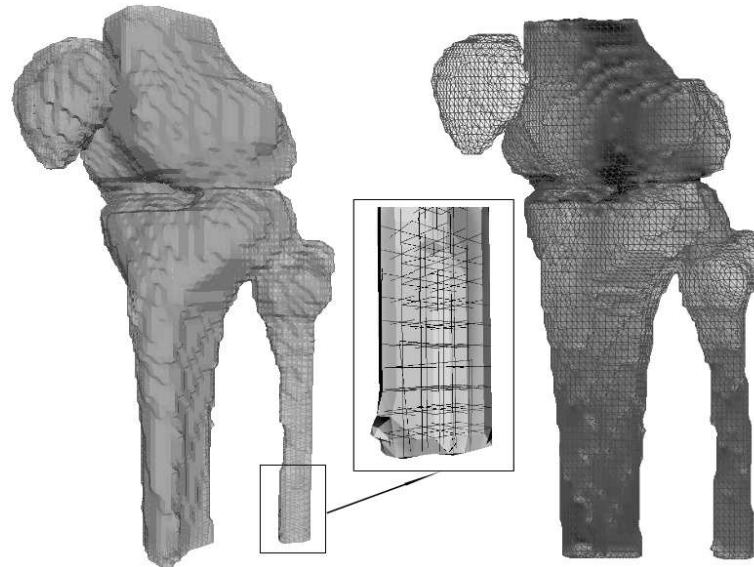


Figure 12: On the left, the mechanical model of the knee joint. The finite element mesh was generated in the same way as that described earlier. The outer 3-node elements are made transparent so that the inner 2-node elements can be seen. On the right, the results of the analysis after applying the same load and boundary conditions as those used in the previous knee study.

6 Summary and Results

In this paper we presented a novel system to help the surgeon in planning orthopedic operations. Using this system we made a virtual biomechanical lab and carried out various FEA studies of a pelvis, a knee, and a wrist. We compared three different finite element mesh types and discussed their advantages and disadvantages. After we created a mixed model and then, using a knee study, we showed that it was faster and requires less memory than the two other models.

The system at present is still in its experimental stage. It is able to perform all the above-mentioned tasks, but there are still parts where some user input or a user decision is required. For example, in the segmentation phase the seed points have to be set manually, and the results should be checked visually to see whether the segmentation meets the user's needs. Then, of course, information exchange with the FEA program is not automatic here. A session file is created by the user during the mesh-generation phase, which will then be read by the FEA program in the analysis phase.

As we said earlier, the MedEdit system has now been implemented and works quite well. In general, it is able to create both the geometric and mechanical models

in about 5 minutes, including the time spent on the manual segmentation. As for the FEA program, it takes about 10 minutes to perform a 3D pelvis volume study (on a PC with a 2-GHz processor and a 1.5 GB RAM).

Our stress results seem to meet clinical expectations, although quantitative tests and measurements still have to be done.

7 Future Plans

As we mentioned in the introduction, the orthopedic-trauma surgeon can generally use only X-ray, CT, MRI images and his own clinical expertise for therapy planning. Mechanical modeling can provide reliable data that ensures the stability of the patient's osteosynthesis prior to the surgical procedure. Certain complications could be avoided if a biomechanical computer modeling approach like ours is used. A detailed and prompt assessment of the surgical problem could be provided with the help of our system.

Our software solution will offer new possibilities for the surgeon. It complements current visual analysis methods and it could also be utilised as a tool in postgraduate education and medical training. We have also created a trainer tool to help students better understand the causes of various bone fractures. Students who are interested can access this system by using our MedSys website [16].

References

- [1] Geiger B. *Three-dimensional modeling of human organs and its application to diagnosis and surgical planning*. INRIA, 1993.
- [2] Varga E, Halmai Cs, Kuba A, Ollé K, Erdőhelyi B. *MedEdit, a Computer Assisted Planning System for Orthopedic-Trauma Surgery*. Proceedings of the 25th International Conference on Information Technology Interfaces, 2003. p. 507-512.
- [3] Nyúl LG, Falcao AX, and Udupa JK. *Fuzzy-Connected 3D Image Segmentation at Interactive Speeds*. SPIE Medical Imaging, 2000. 3979 p.212-23.
- [4] Garland M and Heckbert PS. *Surface simplification using quadric error metrics*. ACM SIGGRAPH' Computer Graphics, 1997. p. 209-216.
- [5] Robb RA, Hanson DP, and Camp JJ. *Computer-aided surgery planning and rehearsal at Mayo Clinic*. Computer, 1996. 29(1):39-47.
- [6] Lorensen WE and Cline HE. *A High Resolution 3D Surface Construction Algorithm*. ACM SIGGRAPH Computer Graphics, 1987. 21, (4).
- [7] Galloway RL, Maciunas RJ. *Stereotactic neurosurgery*. Crit. Rev. Biomed. Eng. 18, 1990. 207-233, (26).

- [8] Nolte LP, Zamorano LJ, Jiang Z, Wang Q, Langlotz F, Berlemann U. *Image-guided insertion of transpedicular screws: a laboratory set-up*. Spine 1995 497-500, (20).
- [9] Mimics at www.materialise.be
- [10] Flashpoint 5000 Stryker/Leibinger (www.boulderinnovatorsgroup.com)
- [11] NDI Optotrak Northern Digital (www.ndigital.com)
- [12] ART/Qualisys ART (www.ar-tracking.de)
- [13] Ascension Laser Bird (www.ascension-tech.com)
- [14] Varga E.: *Biomechanical and Clinical Investigation of Pelvic Ring Injuries*. Annals of the Albert Szent-Györgyi Medical and Pharmaceutical Center, 2000. Vol. 47.
- [15] Beer FB and Johnston ER. *Mechanics of Materials*. McGraw-Hill Inc., 1980. 42.
- [16] Our MedSys system website: <http://www.inf.u-szeged.hu/~medsys>

Received February, 2004