# Preparing Surgical Operation Plans for Finite Element Analysis Using the MedEdit System

**Balázs Erdőhelyi, Krisztián Ollé, Endre Varga, Attila Kuba**

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.

CT images serve as input for the MedEdit system. After the segmentation of the bone from other tissues, a geometric model is built from the 3D volume data. The geometric model consists of a triangle mesh, which is used for fast 3D presentation and navigation. The system provides a variety of tools for planning a surgical operation, like moving, drilling, cutting of bones or inserting implants. To simulate the effects of the planned intervention under specific conditions, finite element analysis is used. To complete the geometric model to a mechanical one, material properties as load and boundary conditions are added to the finite element mesh.

Three types of finite element meshes are investigated and compared. The first takes the triangle mesh of the geometry, and uses 3-node triangular thin shell elements for the analysis. The second model uses 8-node solid finite elements for every voxel in the segmented 3D volume, which enables the simulation of the inner bone tissue and the insertion of implants. The third model combines the previous two by using shell elements for the surface and 2-node elements for the inner structure.

The possibility of inserting implants to connect broken bone pieces is also investigated and presented on different examples (knee joint, pelvic ring and hand).