Introduction: When assessing a fracture, an orthopedist has to integrate multiple factors before deciding on the optimal treatment plan. These include general patient-factors (age, past medical history, current medical status) [4], surgeon-specific factors (availability, experience, support), hospital-specific factors (available fixation devices, support staff), and fracture-specific factors (bone and soft tissue quality, number of fractures). Fracture treatment decisions are particularly critical in patients with poor bone quality, multiple trauma patients, or medically debilitated patients. Treatment approaches for any specific fracture frequently vary widely among orthopedists.

Recent computer-aided preoperative planning software provides visualization, measurement, and implant selection and positioning tools. However, none provides the biomechanical information necessary for surgical procedures of load bearing bones.

We are currently developing a patient specific computer aided quantitative planning and analysis tool that will assist the orthopedist in fracture treatment planning, and improve treatment outcome. The software will assist surgeons in optimal implant type, size and position determination based on biomechanical considerations. The computer aided quantitative planning and analysis tool will be based on simulation of the mechanical response of the patient's affected bone using a Finite Element (FE) model generated from a QCT (quantitative computerized tomography) scan [3]. This paper is the first to describe our approach and preliminary results for long bone fracture treatment.

Goals: The goals of this study are: 1) to describe the requirements and technical elements for practical CAD/CAM orthopedic software to support computer aided
preplanning and computed aided surgery; 2) to establish the validity of the FE model for several different fracture types and fixating devices, and; 3) to present preliminary results comparing the simulated mechanical response with mechanical testing on third-generation saw bones.

**Methods:** The key requirements for the clinical acceptability of patient specific (PS) preplanning and FE technology [3] are: 1) all the planning must be performed by the clinician in a single software environment; 2) the various steps must be as automatic or semi-automatic as possible, with intuitive input from the clinical users; 3) robust and accurate; 4) matches clinical staff skills, and; 5) performed in a reasonable amount of time and with moderate effort.

With these considerations in mind, we propose the following sequence of steps. First, a CT scan of the traumatized bone is acquired (Fig 1a). The dataset is imported to the software, and the surgeon indicates the region of interest. The software automatically creates a model of the bone fragments of interest (both surface and volumetric) and assigns material properties to every volumetric element in the model based on correlations between CT Hounsfield units and bone mass density [1,2,5]. It allows the surgeon to interactively visualize and manipulate the fragments in space, explore fracture reduction options, and select and position fracture fixation devices (Fig 1b). These tasks are performed within the commercial package AmiraDev©. Once the desired reduction and fixation is achieved, the ABAQUS© commercial software package is use for FE analysis and result visualization (Fig 1c).

The software includes custom software, coding of existing algorithms, and development of new ones. The algorithms include: 1) automatic volumetric quality mesh generation based on Delaunay triangulation; 2) automatic material assignment using Zannoni's method [1], and; 3) Boolean operations between surface meshes.

To demonstrate the feasibility of the proposed process and tools, two 3rd generation sawbone femurs were used. One was used as is and the other was cut at the inter-
trochanteric line to mimic common intertrochanteric fractures. Both femurs were then CT scanned (Fig 1a). Preoperative planning included fracture reduction, fine adjustment of fracture surfaces with Boolean operations, and implant positioning of a 6mm diameter metal pin. (Fig 1b). The results were then imported into the FE software ABAQUS. The boundary conditions were set to exactly match the mechanical testing environment: no femur head motion and linear displacement of the medial femoral region along the mechanical axis. Additional constraints were added to model the hard contact between the fracture surfaces of the two bone fragments and the tie conditions between the implant surfaces and the bone surfaces. The results from the FE analysis were validated with an axial loading Instron machine (Fig 1d) for both bones: the whole femur underwent a three point bending test and an axial loading test. The fractured femur was tested for axial loading with a crosshead speed of 0.5 mm/min.

**Results:** The time required to perform the entire planning was about two hours; it is expected to decrease when all the procedure steps are integrated into a single application. The deflection of the mid-section of each femur in the mechanical test exactly matched the deflection of the FE analysis, as this was a controlled parameter in the mechanical setup. Reaction forces on the femur head were measured by the Instron load cell and compared to those computed by FE. The values matched within a 5%-10% range.

**Conclusions:** The potential of a real computer aided preplanning tool is enormous and may also assist in additional clinical oriented subjects such as training, failure mechanism inquiries, medical documentation. The proposed methodology and process has been proven for feasibility since it complies with conditions imposed by the clinical environment and clinicians. In-vivo and ex-vivo experiments are to be conducted on a large scale in order to better estimate compliance with the desired properties of accuracy, robustness, and automation.
References


Figure

(a) CT scan of the third generation saw bone. The femur was cut along the intertrochanteric line. (b) Fracture reduction and implant positioning after surface meshing. (c) Graphical output of the strain field developed in an axial loaded femur. (d) Axial load mechanical testing. The boundary conditions are identical in the mechanical test and the finite element model.