Preventing Contact Convergence Problems in Bone-Implant Contact Models

Sebastian Eberle, Peter Augat

Biomechanics Laboratory, Trauma Center Murnau,
Murnau, Germany

Summary

A major problem, when fixing a fracture of the proximal femur with an osteosynthesis, is the unknown force distribution in the bone and the implant. In order to predict the force distribution in bone and implant a Finite Element Analysis was developed.
The Mises equivalent stresses in a Gamma3-Nail were calculated using the finite element method. The Gamma3-Nail was virtually implanted in a model of the standardized femur, a digital model of an artificial bone, which included the drillings of a regular operation. Three different fractures were compared under the load of the one legged stance. The fractures were located intertrochanteric, subtrochanteric and at the lateral neck of the femur. The FE model included all contacts between implant and bone and between the two fracture fragments.
The subtrochanteric fracture loads the implant the most and results in the highest equivalent stress at a groove surrounding the hole for the lag screw. The intertrochanteric fracture results in a lower stress, also located at the groove. The femur model with the lateral neck fracture has its maximum stress at the lag screw.
The calculations showed, that each fracture results in different stress distributions in the nail with respect to location and magnitude. Reason for that are different contact points and leverages, which influence the distribution of force within the implant.

Keywords

Bone, implant, contact, osteosynthesis, ANSYS Workbench

1. Introduction

A major problem, when fixing a fracture of the proximal femur with an osteosynthesis, is the unknown force distribution in bone and implant. The stresses in the implant are unknown and the question which implant is the right one for a specific fracture in terms of stability and fatigue resistance can not be answered on an objective basis.

The determination of strains and stresses on an intramedullary implant in vitro by experiment is very difficult, in vivo just not possible. Furthermore the exact locations of highest stresses on the implant are unknown. Known are the external forces on the bone from experiments and models [1].

In order to predict the force distribution in bone and implant a Finite-Element-Analysis (FEA) was developed. The purpose of this analysis was to calculate the stresses in an intramedullary implant in three FE models with different fractures in the proximal region of the femur. A special focus was put on the contact modeling of the bone implant interface.

2. Materials and Methods

Fractures of the proximal femur are frequently fixed with intramedullary implants, such as the Gamma3-Nail. The Mises equivalent stresses in a Gamma3-Nail (Stryker Inc.) were calculated using the finite element method (FEM). The Gamma3-Nail, consisting of nail, lag screw and distal locking screw, was virtually implanted in a model of the standardized femur [5], which included the drillings of a regular operation (see fig. 1).

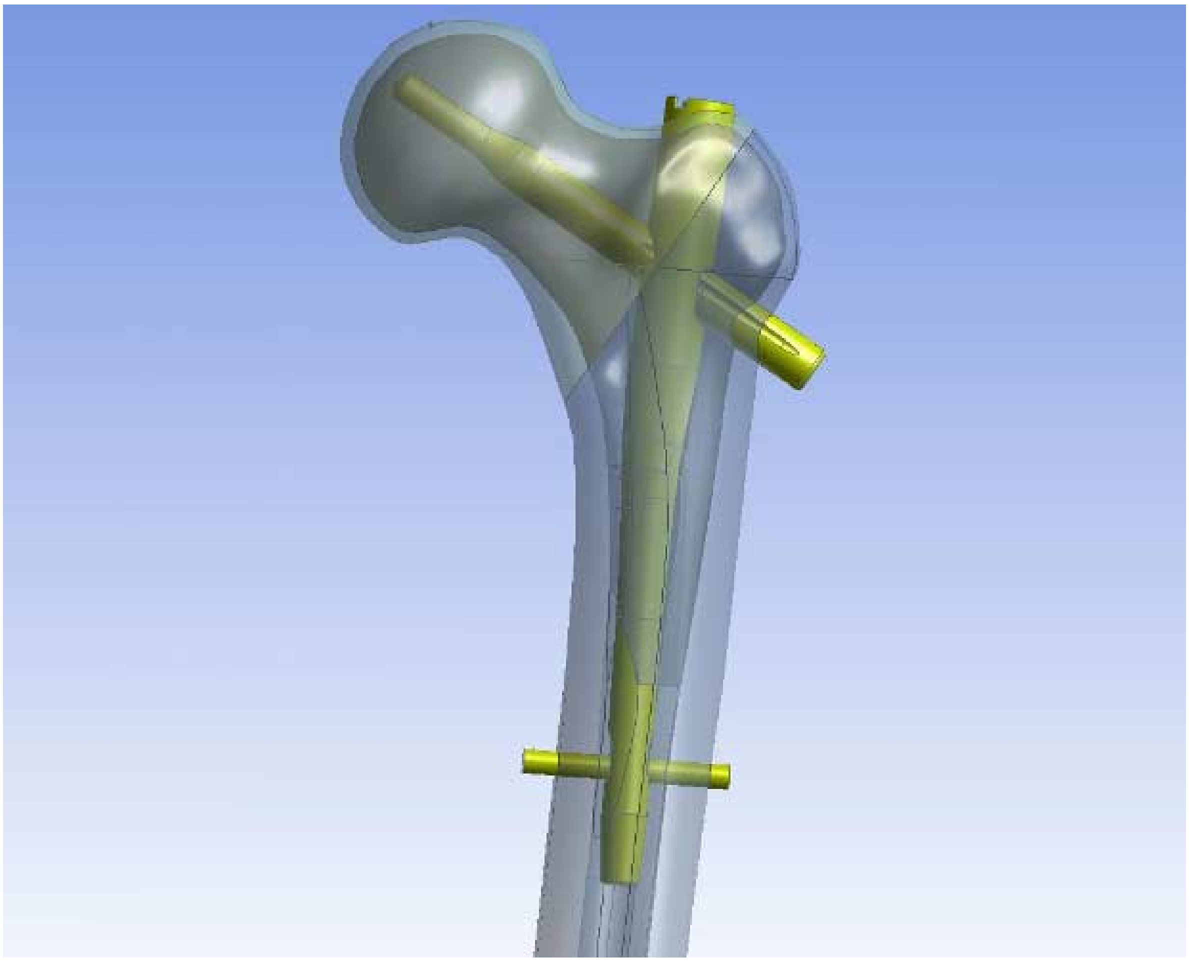

Fig. 1: Virtually implanted Gamma3-Nail (yellow) in the femur model (transparent).

Three different fractures were compared under the load of the one legged stance of a 80kg person, which is a typical load case for the human femur. The fractures were located intertrochanteric, subtrochanteric and at the lateral neck of the femur (see fig. 2).

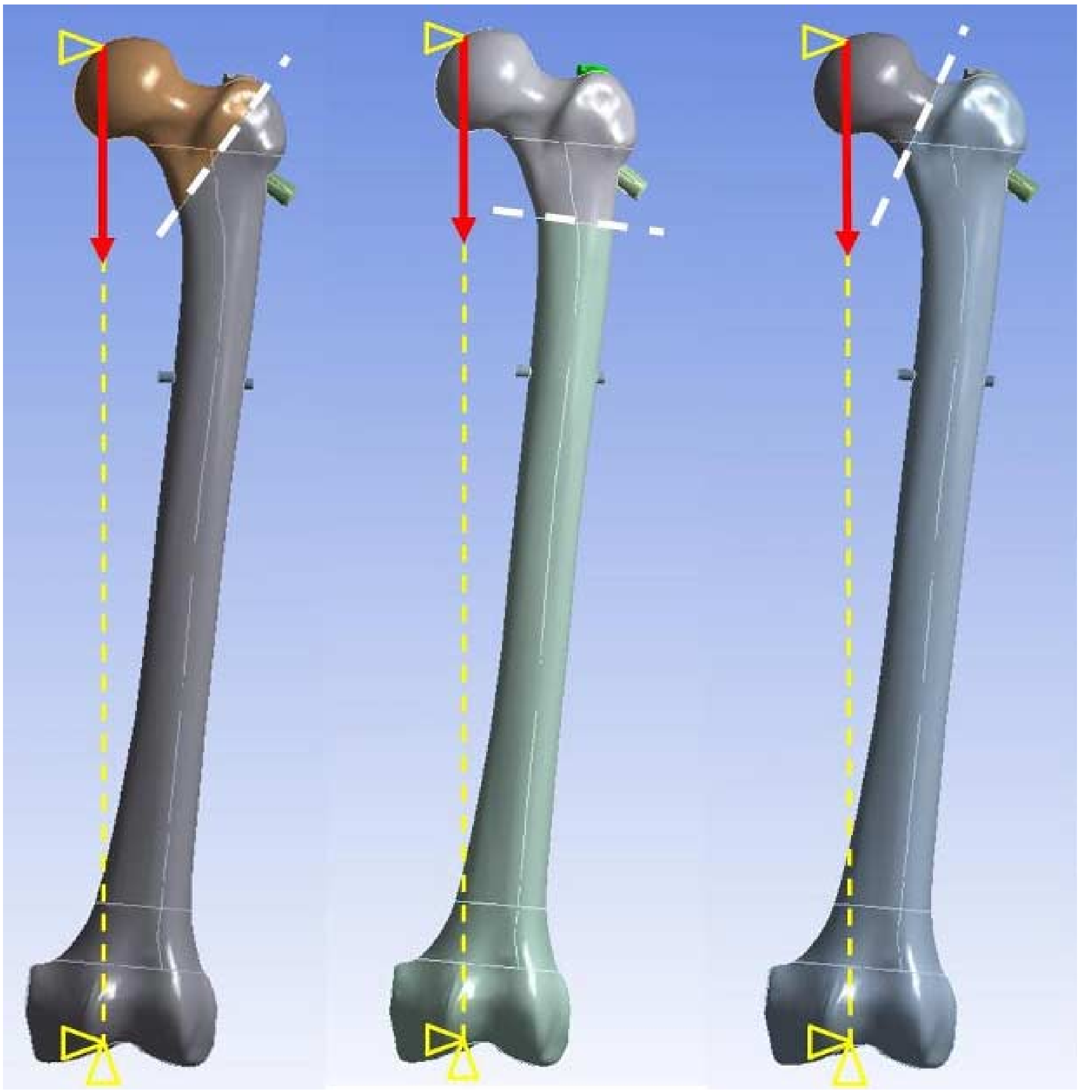

Fig. 2: Three models of the standardized femur with lines of fractures (white), forces (red) and lines of forces and constraints (yellow).

The FE model included all contacts between implant and bone and between the two fracture fragments. For the contact modeling several important points were taken into account:

- Geometry of bone and implant. Where do they penetrate each other and where are gaps between the two contact partners? With an appropriate contact stiffness, a press fit of the nail in the bone can be realized.

- The contact type should be as realistic as possible. In this case all contacts were modeled as frictional contacts except the contacts between the threads of screws and the bone. These were modeled as bonded contacts.

- For frictional contacts a coefficient of friction is to be determined. Coefficients of friction were considered for the contact between Ti6Al4V-Ti6Al4V [3], bone-Ti6Al4V [4] and bone-bone (determined in own experiments) (see Tab. 1).

Table 1: Coefficients of Friction

Materials	Coefficient of Friction
Ti6Al4V-Ti6Al4V	0,38
Bone-Ti6Al4V	0,36
Bone-Bone	0,46

- The Augmented Lagrange formulation was chosen for the contact algorithm. This method results in a better conditioning and is less sensitive to the magnitude of the contact stiffness.

- Most important in contact problem with several unconstrained contact partners is the prevention of rigid body motions. This can be done by the assignment of weak springs to every contact partner with a defined spring stiffness and an adjusted time stepping. So the contact bodies perform a known displacement in the first substep, which is defined by the spring stiffness and the external forces.

The force on the femoral head and the constraint by the hip joint were idealized by a force of 1866N and two constrained translatory degrees of freedom (DOF) normal to the line of force. The line of force went through the femoral head and the pivotal point of the distal end of the femur. This pivotal point is in vivo made up by the knee joint and has three rotatory DOF which were idealized by locked translatory DOF in the FE model (see fig. 2). The FE models had about 25.000 to 50.000 elements and 43.000 to 90.000 nodes and were realized using ANSYS®Academic Research, v.11.0 Workbench™ (ANSYS Inc., Canonsburg, USA).

3. Results and Discussion

Amount and location of the Mises equivalent stresses on the implant differed between the three fractures. The subtrochanteric fracture loads the implant the most and results in the highest equivalent stress (531 MPa) at a groove surrounding the hole for the lag screw. The intertrochanteric fracture results in a lower stress (410 MPa), also located at the groove. The femur model with the lateral neck fracture has its maximum stress (368 MPa) at the lag screw (see Fig. 3). So every fracture distributes the force on the femoral head in another way.

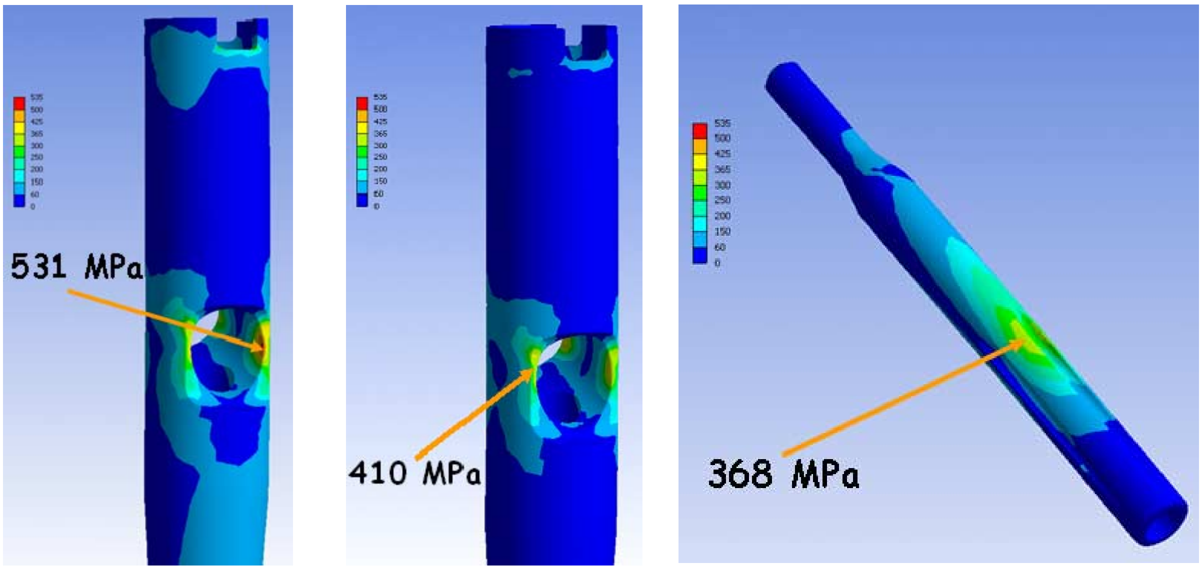

Fig. 3: Max. Mises equivalent stresses on implant from FE models with subtrochanteric fracture (left), intertrochanteric fracture (middle) and lateral neck fracture (right).

Considering, that the Mises equivalent stress is the failure criterion for ductile materials, the calculated equivalent stresses would not result in a breakage of the nail. The used alloy Ti6Al4V has a yield strength of 1020 MPa and a tensile strength of 1190 MPa. But with a fatigue strength of 650 MPa the nail could fail, if cyclically loaded with a force slightly higher than the used 1866N on the femoral head [2].

This study has several limitations. At that time the experiments to validate the FE models are still in progress. The validation is carried out by measurements of strains at crucial spots on the implant. The strains from simulation and experiment will then be compared. Furthermore the Gamma3-Nail is not normally used for the fixation of a lateral neck fracture. But in this case the fracture served well to produce a very dissimilar force distribution in the nail. Furthermore we used just one loadcase in this study and also used plastic bones, which have the mechanical properties of a human bone, but still are dissimilar in some aspects to a real bone.

4. Conclusions

The FE model of the standardized femur offers the possibility to compare different implants and fractures. The calculations showed, that each fracture loads the implant in a different way and results in different locations and amounts of stress in the nail.

So different types of fractures can have strong effects on the loading of a nail. Reason for that are different contact points and leverages, which influence the distribution of force within the implant.

5. References

[1] Bergmann G *et al.*, Hip contact forces and gait patterns from routine activities., J Biomech. 34(7):859-71, 2001

[2] Dindorf C.: Ermüdung und Korrosion nach mechanischer Oberflächenbehandlung von Leichtmetallen, TU Darmstadt, Diss., 2006

[3] Lempert GD, Tsour A, Reduction of static friction between surfaces of Ti-6Al-4V and between surfaces of Ti-6Al-4V and Al-7075, Surface and Coatings Technology 52(3):291-295, 1992

[4] Mischler S, Pax G, Tribological behavior of titanium sliding against bone., European Cells and Materials 3 (Suppl. 1):28-29, 2002

[5] Viceconti M *et al.*, The 'standardized femur program' proposal for a reference geometry to be used for the creation of finite element models of the femur., J Biomech 29(9):1241, 1996