EXPLICIT FINITE ELEMENT SIMULATION OF UNCEMENTED TOTAL HIP ARTHROPLASTY: IMPLANT INSERTION AND LOADING

Marzieh Ovesy, Benjamin Voumard, Philippe Zysset
ARTORG Center for Biomedical Research, University of Bern, Bern, Switzerland

Introduction

The term primary stability is used for evaluating the success level of total hip arthroplasty (THA) and is mainly quantified as the amount of micromotions occurring at the bone-implant interface. Finite element analysis (FEA) has been used to measure the amount of micromotions around the implant during loading. However, these models have not considered the damage induced in bone throughout implantation and the complex material representation of bone [1]. This study aimed to evaluate the effect of implantation on bone in terms of stress and quantify the irreversible micromotions produced during subsequent loading cycles.

Materials and Methods

A cadaveric right femur (age: 87, gender: female) was scanned using quantitative computed tomography (QCT). Using a pre-operative surgery planning, the Optimys (Mathys Medical, Switzerland) femoral stem was positioned in the bone. The head of the femur was cut accordingly. To simulate the canal opened by the rasping stage, the implant image was shrank by 2 mm and subtracted from the femur. Then, the femur was meshed using tetrahedral elements and the corresponding bone volume fraction (BV/TV) was assigned to each element (Medtool, Austria). The finite element simulation was performed in Abaqus (Dassault Systems, France) using an explicit scheme. The material model was chosen to represent the bone as a BV/TV dependent homogenized material having elastic-plastic behavior based on a quadric yield surface [2]. Damage in this model is a continuous function based on the cumulative plastic strain in the material and element deletion is triggered when the elements are fully damaged [3]. The implant was assumed to be rigid. The interaction between the bone and implant was modelled as unilateral contact with a friction coefficient of 0.3. The implantation was simulated by a displacement-based loading protocol defining the position of the implant tip along the femur shaft, while all other degrees of freedom were free. Finally, the implant undergoes a loading cycle representing a gait cycle from the Orthoload database. The irreversible micromotions were evaluated as the relative motion between bone and implant at the end of each loading cycle [4].

Results

The implantation was simulated successfully. The maximum von Mises stress after the implantation was 85.5 MPa and dropped to the value of 75.7 MPa after loading. The maximum normal and tangential micromotions after the loading stage were 0.6 and 1 mm, respectively both showing the highest values at the distal section of the implant showed in Figure 1. e,f. However, the medians were 0.15 and 0.28 mm for normal and tangential cases. Due to existence of tangential values, it is inferred that the friction value affects the micromotions.

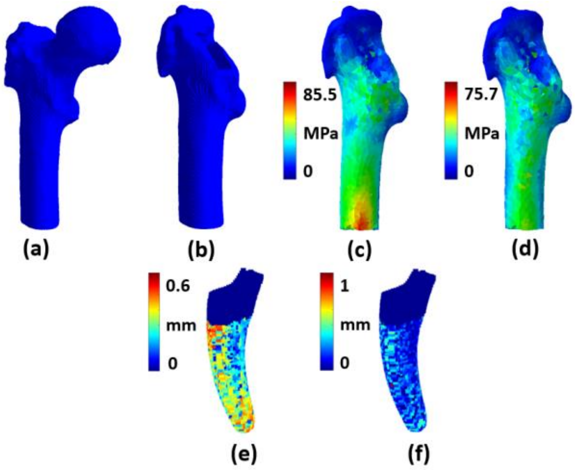

Figure 1: Intact femur (a). Pre-implanted femur shape (b). von Mises stress distribution after implantation (c) and after loading (d). Tangential (e) and normal (f) distribution of the micromotions.

Discussion

The developed approach based on explicit FEA and bone damage allows for the first time to model the hip implantation process and evaluate primary stability after implantation and migration of the implant after the first physiological loading cycle. It also provides the opportunity to quantify the stresses induced by implantation and apply loading protocols on the damaged femur rather than on an intact structure. The amount of micromotions would be quantified resulting in a better understanding of the primary stability level. This first proof of principle opens the path to investigate the effect of press-fit and friction and on the primary stability of THA.

References

1. Viceconti et al, J Biomech, 33 1611-1618, 2000.
2. Schwiederzik et al, JBMM, 12(6):1155-1168, 2013.
3. Ovesy et al, J Biomech, submitted, 2019.
4. Bergmann et al, J Biomech: 34(7) 859-71, 2001.

Acknowledgements

Funding of RMS Foundation (grant no E16_0001/HOM-FEM) and support of Mathys Medical for providing the implant CAD are gratefully acknowledged.