PATIENT-SPECIFIC FINITE ELEMENT MODEL FOR INTRAOPERATIVE FRACTURES PREDICTION WITH A COMMERCIAL DESIGN

Maila Petrucci (1,2), Antonino A. La Mattina (1,2), Cristina Curreli (1,2), and Marco Viceconti (1,2)

1. Department of Industrial Engineering, Alma Mater Studiorum - University of Bologna, Italy;
2. Medical Technology Lab, IRCCS Istituto Ortopedico Rizzoli, Bologna, Italy

Introduction

Total hip arthroplasty (THA) is one of the most established surgical procedures. However, many are still causes of failure that a prosthesis implant could encounter, among which is the Intraoperative Femoral Fracture (IFF). Even if it does not represent one of the most frequent complications, one important aspect is that the failure incidence is strongly linked to the stem design. For this reason, investigating this failure scenario is essential for companies that want to put a new device on the market. In the last years, Finite Element (FE) models, and in general terms *in silico* trials, are increasingly becoming a tool to support experimental tests for the analysis of the performances of new devices. A new approach for predicting IFF using FE models capable of simulating crack propagation was recently proposed [1]. However, mock implant designs were used in this conceptual study. The present work aims to propose a real-world use of the workflow previously developed, by applying it to the Avenir Complete Hip System (Zimmer Biomet), identified from clinical register data managed by the Rizzoli Orthopaedic Institute in Italy as the one with the lowest IFF incidence.

Materials and Methods

A subject-specific FE model was developed starting from the CT scan of one patient: 3D femur geometry was obtained with a threshold-based segmentation, and a Boolean subtraction with the rasp of the nominal implant size was performed. The stem size and pose were obtained from pre-operative planning performed by an expert surgeon. The femur and stem components were meshed separately, and the material properties were then assigned: the bone was mapped element-wise using Bonemat software [2], while Titanium Alloy properties were assigned to the stem. A compressive load with a stepped curve from 1000 to 10 000 N was applied along the stem axis, and the femur was constrained 50 mm under the stem apex (Fig. 1a). A maximum principal strain criterion was used to identify local element failure. The elements that exceeded the tensile strain threshold value of 0.0073 [2] were deactivated, *i.e.*, their elastic modulus was imposed as quasi-zero; after element deactivation, the load was kept constant to allow stress redistribution. From the outputs of the FE analysis performed in Ansys MAPDL, two main quantities were considered to identify and characterise the IFF: the volume ratio between the deactivated elements and the femur, and the area of the internal crack surface.

Results

Results obtained from the FE analysis in terms of crack propagation are shown in Fig. 1b. Two fracture paths, observed in the proximity of the lesser trochanter, reached the external femur surface under an applied load of 3000 N. The crack computed final volume ratio and area were 0.56% and 856.58 mm^2, respectively, showing an increasing trend as soon as the crack propagated.

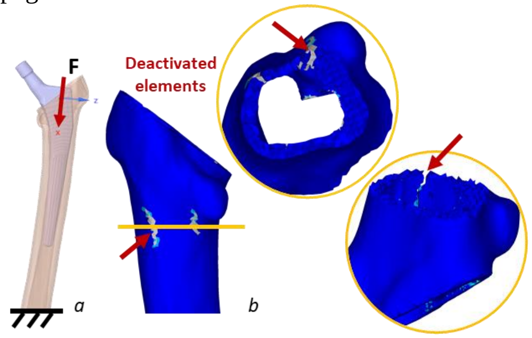

Figure 1: Boundary conditions defined for the IFF simulation (a) and crack propagation up to the external surface (b).

Discussion and conclusions

This study demonstrated the possibility to apply the workflow recently developed to simulate IFF using a commercial stem design for which the clinical output and IFF incidence are known. The results obtained confirmed the validity of the metrics proposed to define the damage quantification criteria. In the next future, the primary stability of the commercial stem will be analysed, considering that, from the previous study with a non-commercial stem, the push-out stiffness rapidly decreased as soon as the crack started to propagate. Once the simulations will be applied to an augmented virtual cohort and the digital twin solution will evolve into an In Silico Trial, it will be possible to clinically validate the methodology. The present work represents a step forward towards the development a computational environment that estimates the IFF failure risk for THR.

References

1. Petrucci M. et al., Abstract 27th ESB Congress, Porto, 2022.
2. E. Schileo et al., J Biomech., 41(2):356–367, 2008.

Acknowledgements

The authors would like to thank the team of Prof. Francesco Traina for the pre-operative planning, and Zimmer Biomet Corp. for providing the CAD models. This study was supported by European Commission through the H2020 CompBioMed2 (grant ID 823712).