PROXIMAL FEMUR FRACTURE RISK ASSESSMENT USING FINITE ELEMENT METHOD SIMULATIONS

Ariel F. Pascaner (1), Francesca Rotini (1), Stefania Marconi (1), Amir Sternheim (2)(3), Zohar Yosibash (4), Alessandro Reali (1), Michele Conti (1), Ferdinando Auricchio (1)

1. Civil Engineering and Architecture Department, Università degli Studi di Pavia, Pavia, Italy; 2. National Unit of Orthopaedic Oncology, Tel-Aviv Medical Center, Tel-Aviv, Israel; 3. Tel-Aviv University, Sackler Faculty of Medicine, Ramat Aviv, Israel; 4. Computational Mechanics and Experimental Biomechanics Lab, School of Mechanical Engineering, The Iby and Aladar Fleischman Faculty of Engineering, Tel-Aviv University, Ramat-Aviv, Israel.

Background

Bones' tumor metastasis is a disease with very high incidence (approximately one third of all cancers), being the proximal femur the most common site. Osteolytic metastases may cause pathological fractures of the bone, in which case life expectancy is less than one year following the fracture [1]. This work aimed at developing a framework for the assessment of patient-specific risk of fracture using finite element method (FEM) simulations, which would assist in surgical planning and optimized prosthetics design.

Materials and methods

Left and right femurs of a female patient (age: 57 years old, weight: 60 kg) with a proximal bone tumor were automatically segmented from the CT scan using Simfini Software [2] (Figure 1, left). High-order volumetric tetrahedral meshes were generated from the segmentations (approximately 70k elements each) and material properties were computed from the CT Hounsfield Units (Figure 1, center). For each geometry, we performed a FEM simulation with the Mechanical APDL modulus of Ansys 2022 R2, using the following boundary conditions. The most distal slice of the mesh was fixed by imposing 0 displacement on all the degrees of freedom of all nodes. A force was applied on the femoral surface in the direction defined between the head center and the intercondylar notch. The magnitude of the force was computed as 2.5 times the weight of the patient (1470 N) and it was distributed among the nodes within an 11 mm diameter circle on the top of the femoral head. Maximum total displacement and maximum compression strain (E3) were assessed, being the latter an indicator of fracture risk.

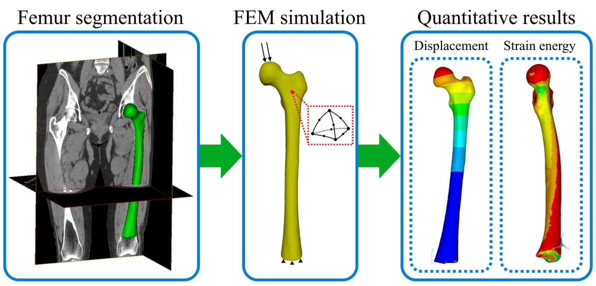

Figure 1: Workflow consisting of: femur automatic segmentation from the CT scan, generation of the 3D high-order tetrahedral mesh and FEM simulation to obtain quantitative results.

Results

Simulations were performed using an Intel Core i7-11370H processor @3.30 GHz, with 32 GB of RAM. Table 1 shows the maximum total displacement and maximum compression strain (E3) for the left and right femurs. An increased maximum compression strain was observed on the right femur (-4414 µS vs -3863 µS), corresponding to the location of the tumor, evidencing a high risk of fracture. The obtained values are consistent with the literature [3], where *ex-vivo* validation of the FEM simulations was conducted.

Femur	Max. displacement [mm]	Max. compression strain (E3) [µS]
Left	5.58	-3863
Right	5.84	-4414

Table 1: Maximum total displacement and compression strain (E3) obtained for both femurs.

Conclusion

In this work, we performed FEM simulations of the femurs of a patient with bone cancer, showing an increased compression strain on the tumorous femur, which is a sign of high fracture risk. Adequate surgical planning is crucial in these types of tumors, given the short survival after pathological fractures. The information extracted from a CT scan allows the generation of 3D bone models, which can be then used to simulate virtual implantation of prosthetics, assisting surgeons in the decision-making process. Further work should be done to achieve optimized patient-specific prosthetics design, allowing the fabrication of long lasting implants.

References

1. M. J. Parker et al, Hip Hint: J Clin Exp Res Hip Pathol Ther, 21(5):526-530, 2011.
2. Z. Yosibash et al, Comput Math, 80(11):2417-2432, 2020.
3. Y. Katz et al, Clin Biomech, 77:1-11, 2020.

Acknowledgements

This work was supported by "AIO – AI for Orthopedics" project, funded by MAECI (Italy) and MOST (Israel).