10th International Congress
of Croatian Society of Mechanics
September, 28-30, 2022
Pula, Croatia

10^{th}ICCSM

Comparison of the conventional loading case on femur with Pauwels type III fracture with force reduction loading: A finite element study

Petra Adamović*, Nina Bočkaj*, Tanja Jurčević Lulić*, Janoš Kodvanj*

**University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Department of Applied Mechanics, Ivana Lučića 5, Zagreb, 10000, Croatia*
E-mails: {petra.adamovic,tanja.jurcevic.lulic,janos.kodvanj}@fsb.hr, ninabockaj@gmail.com

Keywords: Pauwels type III, finite element method, force reduction, inverted triangle configuration

1. Introduction

The application of finite element method in interdisciplinary sciences such as biomedical engineering has been quite common in the last few decades [1]. Treatment of intracapsular hip fractures such as femoral neck fracture of Pauwels type III still remains a challenging matter, given the fact that this disease nowadays involves more than 50% of all proximal femoral fractures [2]. While creating an internal fixation device for that matter, it is necessary to conduct experimental tests. The goal of simulations is to select the crucial device test loads and setups, i.e., to reduce the cost and time consumption of the tests. While most authors use a very primitive form of model loading in their simulations [3-5], this study uses two types of femur's loading cases (climbing stair and walking) and compares two mechanically equivalent force loadings with their amounts.

2. Materials and methods

A left, 4th generation composite femur (Sawbones®, USA) was scanned using computed tomography and its geometry was obtained using Mimics software (17.0, Materialise NV, Belgium). A 3D model of the femur with three screws in an inverted triangle configuration was created in SolidWorks software (2020, Dassault Systèmes, USA). The screws' material was set to be titanium with a radius of 5 mm. All materials were defined as linearly elastic, homogeneous and isotropic, while their properties and interactions are listed in Table 1.

Table 1. Model properties and boundary conditions in Abaqus

Material	Elastic modulus [GPa]	Poisson's ratio [-]	Boundary condition
Screws (titanium)	110	0.3	With all other models: Tie
Cortical bone	17		On fracture site: surface-to-surface (friction coefficient: 0.2)
Trabecular bone	1.1		

One loading case imitated stair climbing and the other one walking. Both loading cases [6] were divided into a four-point (P1, P2, P3, P4) loading case and in a reduction of those forces in one point (A) on the femur. These were further divided into models with all three force components and the ones with resulting forces and moments only. That resulted in eight simulations made in Abaqus software (6.14-5, Dassault Systèmes, France), schematically shown in Fig. 1.

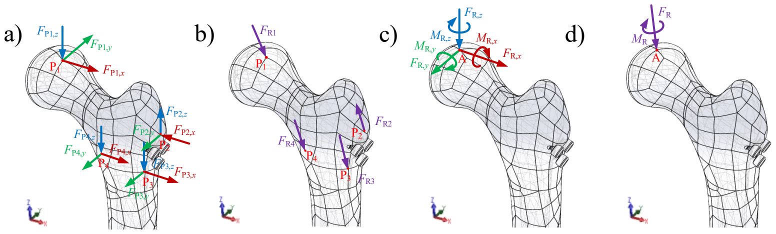

Figure 1. Mechanically equivalent loading forces for both stair climbing and walking

3. Results

Fig. 2 shows the distribution of maximum von Mises stress on elements and displacements of nodes on femur, while Fig. 3 displays the relative fracture displacement (displacements of node pairs) distribution along the upper side of the fracture for stair climbing and walking.

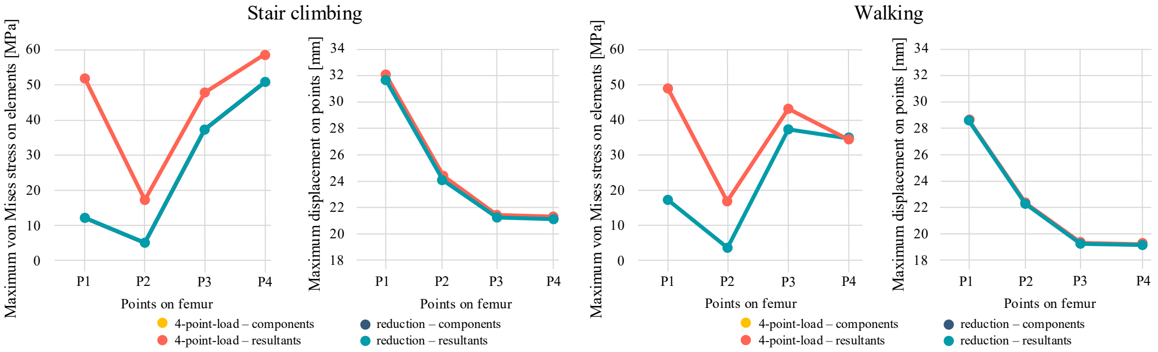

Figure 2. Maximum von Mises stress distribution and displacements for the two load cases

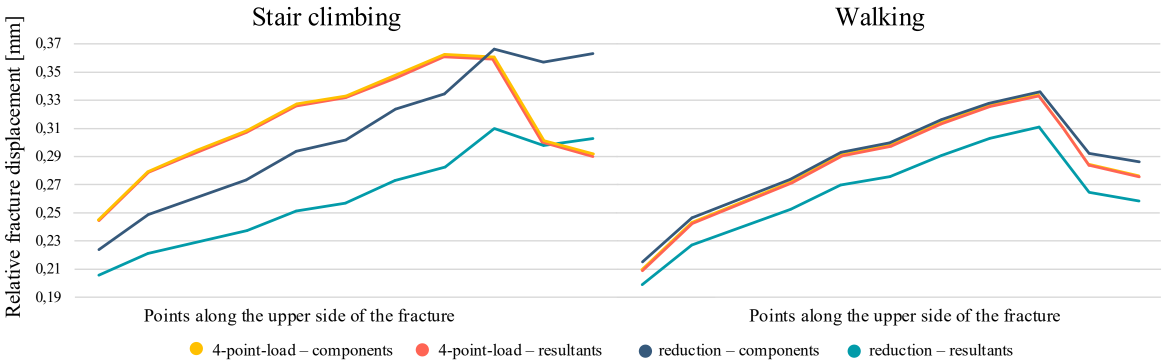

Figure 3. Relative fracture displacement for the two load cases

4. Conclusions

While designing implant devices, multiple loads with various points on femur should be used to determine the maximum von Mises stress values and its distribution. The results further show that almost equal displacements (max. error of 0,52%) are present in all loading cases, thus indicating that this approach of force reduction can be used in the examination of fracture displacements.

References

[1] Taylor, M., Prendergast, P.J. Four decades of finite element analysis of orthopaedic devices: Where are we now and what are the opportunities? *J biomech*, 18;48(5):767-78, 2015.

[2] Ridzwan M.I.Z., Sukjamsri C., Pal B., van Arkel R.J., Bell A., Khanna M., Baskaradas A., Abel R., Boughton R Cobb J Hansen U. Femoral fracture type can be predicted from femoral structure: a finite element study validated by digital volume correlation experiments. *J Orthop Res,* 36(3);993-1001, 2018.

[3] Li, J. Zhao, Z., Yin, P., Zhang, L., Tang, P. Comparison of three different internal fixation implants in treatment of femoral neck fracture – a finite element analysis. *J Orthop Surg Res*, 14:76, 2019.

[4] Corderio, M., Caskey, S., Frank, C., Martin, S., Srivastava, A., Atkinson, T. Hybrid triad provides fracture plane stability in a computational model of a Pauwels Type III hip fracture. *Comput Methods Biomech Biomed Engin*, 476-483, 2019.

[5] Huang, H., Feng, Z., Wang, W., Yang, C., Liao J., Ouyang, J., Finite Element Analysis of Femoral Neck Fracture Treated with Bidirectional Compression-Limited Sliding Screw. *Med Sci Monit*, 30;27:e929163, 2021.

[6] Zeng, W., Liu, Y., Hou, X. Biomechanical evaluation of internal fixation implants for femoral neck fractures: A comparative finite element analysis. *Comput Meth Prog Bio*, 196:105714, 2020.