3D Dynamic finite element simulation of a cementless custom made prosthesis insertion into the femoral cavity

Asiminei AG[1], Pastrav LC[1], Jaecques SVN[1], Mulier M[2], Van der Perre G[1]

[1]*Division of Biomechanics and Engineering Design, K.U.Leuven, Belgium, E-mail: aidageorgeta.asiminei@student.kuleuven.be, cezar.pastrav@mech.kuleuven.be*

[2]*Division of Ortophaedics UZ Pellenberg, K.U.Leuven, Belgium*

Introduction

The long term clinical success of porous coated cementless total hip replacement (THR) depends upon bone growing into the surface of the implant. An adequate bone ingrowth requires both implant stability and a good contact between bone and implant surface. A good initial stability can be achieved by press fitting the stem into the femoral cavity. During the surgical insertion of the prosthesis high stresses occur in the surrounding bone and sometimes the stress values are large enough to compromise the integrity of the bone and consequently to compromise the clinical outcomes of the procedure. Excessive press-fitting of a THR femoral component can cause intra-operative fractures with an incidence of up to 30% in revision cases[1]. The hip stem stability and the stress distribution are related to the contact properties at the bone-implant interface. The interface stresses contribute to the implant stability and to bone growth stimulation. When the mechanical stimulation is not adequate (e.g. because of stress shielding), reduction of bone mass and/or density may be the consequence. For these reasons it is important to identify the contact areas, to observe how the contact areas evolved during the stem insertion, and to understand load transfer mechanisms since these are important factors that can affect bone ingrowth.

Considerable efforts have been made to improve the life expectancy and stability of these implants, and to minimize the stress shielding effect.

The custom made prostheses (CMP) are designed to fit and fill as much as possible the femoral cavity and as a consequence an optimal contact and stability can be obtained[2].

This paper presents a 3D finite element model and the corresponding transient dynamic analysis performed to observe the evolution of contact surface, stress and strain distribution during the insertion of a cementless CMP.

Materials and methods

The femur finite element model is based upon CT scan data of a standardized composite femur (Sawbone) and the CMP finite element model was started from a Standard Triangulation Language (STL) file of an average custom made stem provided by courtesy of Advanced Custom Made Implants S.A./N.V. Belgium. The models were initially designed using the Materialise software package (Mimics® and Magics®). The femoral cavity was obtained applying a boolean operation that extracted the prosthesis volume from the femur, thus the femoral cavity and the prosthesis stem have the same shape and size.

The solid mesh was realized in MSC Patran® using tetrahedral elements with 2 mm edge size.

Using MSC MARC Mentat® the material properties, initial and boundary conditions were defined. The cortical bone was simulated using the corresponding mechanical properties for two external layers of tetrahedral elements (10364). For the remaining elements (13696) of the femur the trabecular bone mechanical properties were used. The stem model is composed of 13287 tetrahedral elements.

An initial velocity was imposed to the prosthesis stem; the prosthesis was constrained to have a displacement along the longitudinal axis of the femur (O-z axis) equal to the dimension of the cavity. The femur was fixed at the level of the condyles.

All materials in the FE model were modelled as linear elastic and isotropic, which is acceptable for the presented type of study[3]. Table 1 lists the assigned properties.

Material properties	Trabecular bone	Cortical bone	CMP (Ti6Al4V)
Young's modulus [MPa]	389	16200	100000
Poisson ratio	0.3	0.36	0.32
Mass density [g/mm3]	0.0005	0.00199	0.00443

Table 1: Material properties

The influence of the presence of friction at the bone-implant interface on the contact evolution was investigated running two different types of analysis (frictionless contact and contact with Coulomb stick-slip friction). The friction coefficient value (0.42) was taken from literature[4]. The prosthesis was inserted into the femoral cavity in 50 steps of equal displacement for both cases.

In the initial stage of the analysis there was no contact between the stem and bone (figure 1), thus the prosthesis could move freely into the femoral cavity until the best position and the best contact was reached in the final stage. The analyses were run in MSC MARC Mentat®.

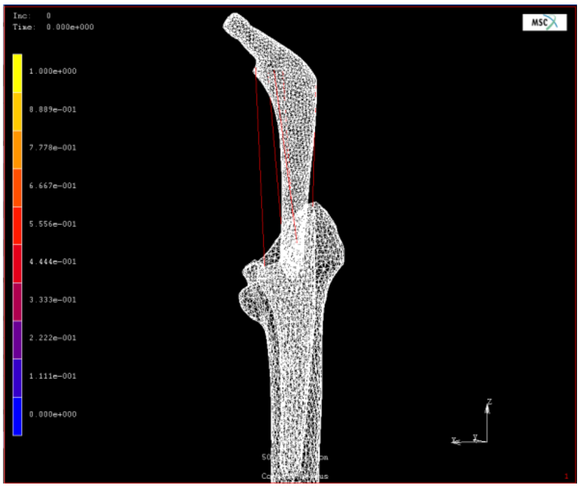

Figure 1: Initial stage of the insertion – no contact

Results and discussion

During the insertion the prosthesis does not perform only a translation in the O-z direction but it tilts and rotates as result of contact reaction forces finding the minimum resistance way to the end point. Due to the irregular stem shape the total contact surface and the contact zone distribution showed irregular evolution during the insertion.

The contact distribution at an intermediary insertion stage is represented in figure 2.

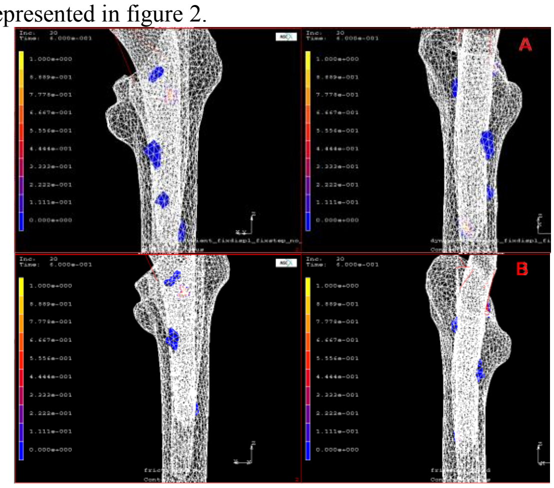

Figure 2: The contact zones – step 30 of insertion (A – frictionless contact, B – contact with friction) (colored zones)

However in the final stage the stem reached an almost complete fit and fill condition (figure 3).

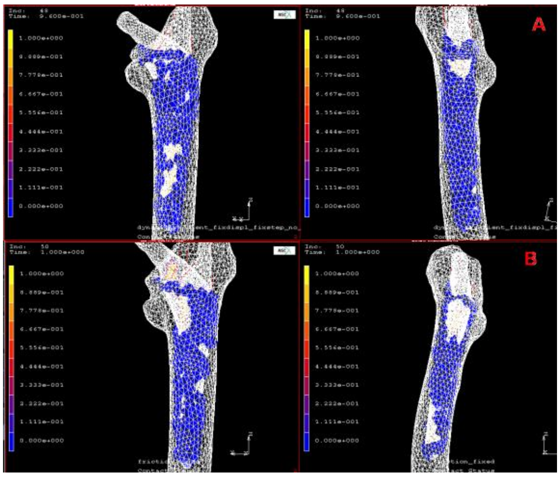

Figure 3: The contact zones – end of insertion (colored zones) (A – frictionless contact, B – contact with friction)

The presence of friction increases the stress values compared with the case of frictionless contact (figure 4).

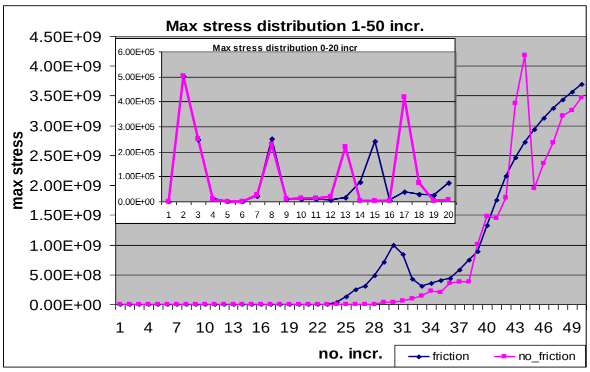

Figure 4: Maximum stress [Pa] during the insertion steps (frictionless contact and contact with friction)

The maximum stress values (figure 4) are related to the probability of the local cancellous bone failure; this means that cracking could occur during the surgical insertion procedure.

The results of the analysis show the presence of large assembly strains during different steps of insertion; these resulted strains must not be ignored.

During the insertion the stress and strain distribution is related to the contact zones. The most important stress occurs in the proximal part of the femur (figure 5).

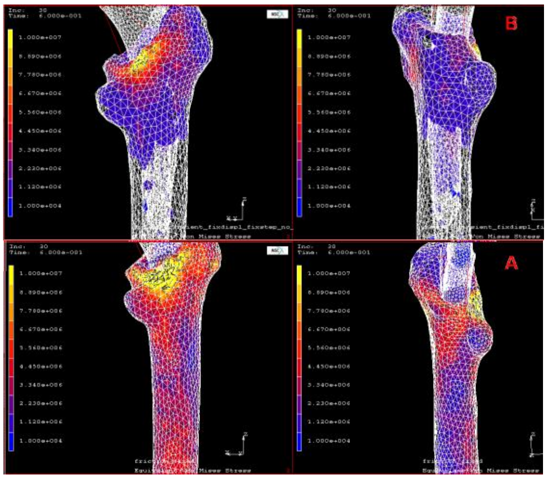

Figure 5: Von Misses stress distribution at an intermediary step of insertion (colored zones) (A – frictionless contact and B – contact with friction)

Conclusions and future work

The CMP design satisfies the requirements to diminish the stress shielding effect by transferring most of the load through the proximal part of the femur.

The total bone-prosthesis contact surface is much larger in the final stage than in an intermediary stage of insertion. A supplementary displacement of the prosthesis will increase dramatically the maximum equivalent stress in the femur. As a consequence it is very important to detect precisely the insertion end point during the surgery.

In future work, the obtained results will be used to study the vibrational behaviour and the mechanical stability of the stem-femur system at intermediary stages and at the end point of insertion. The numerical results will be corroborated with a previous experimental study[5]. Also the influence of the stem shape on the primary stability of the implant will be studied.

References

[1] Meek R.M. et al., Intraoperative fracture of the femur in revision total hip arthroplasty with a diaphyseal fitting stem, *J. Bone Joint Surg.* [Am], 86A(3):480-5, 2004.

[2] Mulier J.C. et al., , Intraoperative production of femoral prostheses, *Joint replacement – State of the Art*, Orthotext, 163-169, 1989.

[3] El'Sheikh H.F. et al., Finite element simulation of the hip joint during stumbling: a comparison between static and dynamic loading, *Journal of Materials Processing Technology*, 143–144 249–255, 2003.

[4] Shirazi-Adl A. et al., Experimental determination of friction characteristics at the trabecular bone/porous coated metal interface in cementless implant, *J. Biomed. Mater. Res.* 27: 167-175, 1993.

[5] Pastrav L.C. et al., An intra-operative vibrational method for hip stem insertion endpoint detection and stability assessment - a pilot study, *J. of Biomechanics*, 39-1: 122, 2006.

Acknowledgements
This research was partially funded by a grant from the K.U.Leuven research council (OT/03/31).