Stress Analysis of Osseointegrated Transfemoral Prosthesis: A Finite Element Model

Ming Zhang[a], Xiang Dong[a] and Yubo Fan[b]
[a].Jockey Club Rehabilitation Engineering Center, The Hong Kong Polytechnic University
[b].Department of Bioengineering, Beijing University of Aeronautics &Astronautics

Abstract

In this study a three dimensional finite element model was established to analyze the interfacial stress distribution in the osseointegrated transfemoral prosthesis. The effects of implant length and screw pitch on the stress and strain distributions were analyzed in terms of bonded and contact bone-implant interfaces. The influences of the contact parameters, contact stiffness and friction coefficient, of the model were evaluated. The results show the differences on the stress distributions at the bone-implant interface in the models with different implant lengths and thread pitches. The highest stress in the model with contact interface is higher than that in the bonded model.

Conclusion

This study presents a FE model for predicting the stress distribution at the bone-implant interface in the application of osseointegrated transfemoral prosthesis. However, some simplifications on loading and boundary conditions are adopted, so there is a need for further validation by experiments.

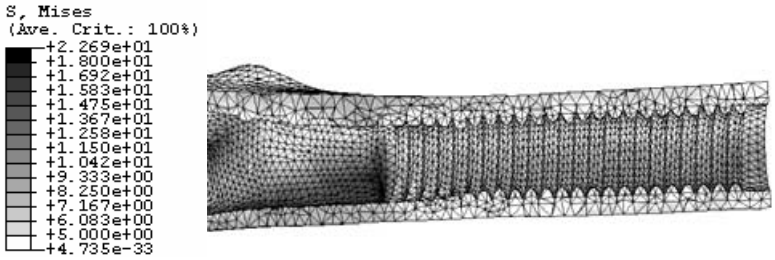

Fig. 4a. Stress distribution on the interior surface of femur in bonded model.

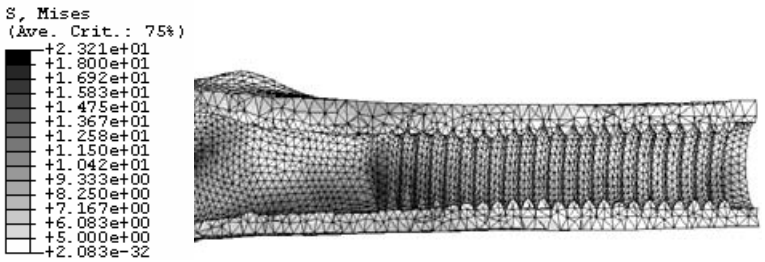

Fig. 4b. Stress distribution on the interior surface of femur in contact model.

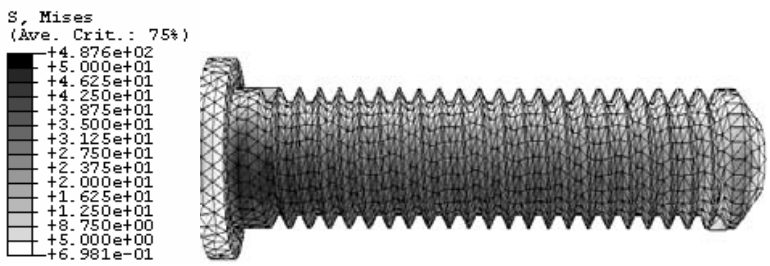

Fig. 5a. Stress distribution on medial side of implant in bonded model.

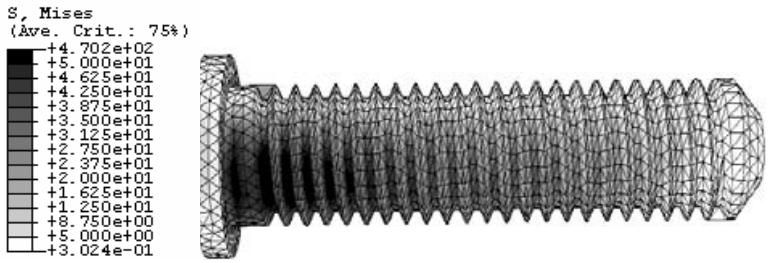

Fig. 5b. Stress distribution on medial side of implant in contact model.

PMID: 17281124 [PubMed - in process]