A FINITE ELEMENT ANALYSIS OF FEMORAL STRESSES IN A SIMULATED FALLING ON THE HIP CONDITION

Miranda N. Shaw [1], Dan M. Dragomir-Daescu [1], Mark E. Bolander [2], Kevin E. Bennet [1], Sean McEligot [1], Michael J. Burke [1], and Geraldine K. Bernard [1]

[1] Mayo Clinic Division of Engineering, Rochester, MN, USA
[2] Mayo Clinic College of Medicine Orthopedic Research, Rochester, MN, USA
E-mail: dragomirdaescu.dan@mayo.edu

INTRODUCTION

With increased incidence of hip fracture risks in osteoporotic patients, it is important to develop preventive treatment options including bone strengthening implants. Analytical and experimental techniques are used to optimize material and design of orthopedic implants. In the current study, the finite element (FE) method was used to evaluate the stress distributions in the proximal region of a femur in a simulated falling on the hip condition. The model used is the standardized femur introduced by Viceconti et al. The simulations were performed using the original, intact model as well as models with implants made of PMMA, Cortoss, and 316L stainless steel.

METHODS

Numerical simulations (FE) using cylindrical implants integrated in the neck region of the standardized femur model (Viceconti 1996) were performed (Figure 1). The analyses were performed using ANSYS 11.0 finite element software to simulate the falling on the hip condition.

Polymethyl methacrylate (PMMA) has been successfully used to augment vertebral bodies, but has several shortcomings including a short working time, radiolucency, and large heat generation (80-124°C) (Erbe 2001). Cortoss synthetic bone filler has been developed more recently to overcome these disadvantages, primarily less heat generation (58-68°C), and provide superior bone integration than PMMA (Erbe 2001, Heini 2003). Also, type 316L stainless steel is widely used in a variety of orthopedic implants.

In the current work, three implant designs were evaluated. The implant was modeled as a cylinder 1, 1.5, and 2 cm in diameter and approximately 8.5 cm in length (Figure 1). Implant material properties were simulated using Simplex P for PMMA and the reported material properties for Cortoss and type 316L stainless steel, respectively. Literature data for femoral bone mechanical properties were used (Table 1). To more closely mimic the anatomical femur, the cortical shell and trabecular bone were modeled. The trabecular bone was modified from the standardized femur to create a hollow femoral shaft.

Boundary conditions were applied to the model to simulate falling on the hip. A frictionless support was applied to the femoral head, and forces of 1000N, 4000N, and 8000N were applied to the greater trochanter. The distal femoral shaft was constrained allowing only rotation perpendicular to the femoral shaft, as shown in Figure 1. The femur was meshed with tetrahedral elements and the implant meshed with hexahedral elements. Meshes with around 900,000 elements were used.

Table 1: Material property data for the standardized femur model and implants.

	Young's Modulus (GPa)	Poisson's Ratio
Cortical	14.2	0.30
Trabecular	1.0	0.30
PMMA	2.828	0.29
Cortoss	5.8	0.29

RESULTS AND DISCUSSION

Maximum von Mises stresses in the femur were recorded in the subcapital and intertrochanteric neck regions for all implant designs. Maximum stresses occur in the cortical bone and decrease as implant diameter increases. Bone stresses are greater with the PMMA implant than the Cortoss and 316L implants. Therefore, it can be deduced that trabecular bone will be stressed more with a less stiff implant such as PMMA and Cortoss than with a stiff metal implant.

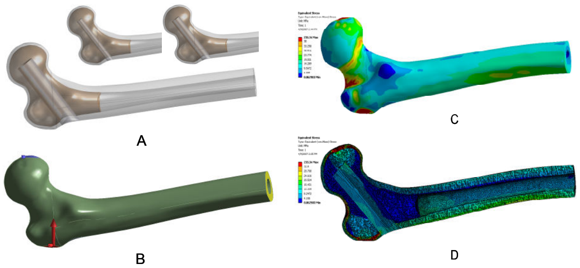

Figure 1: (A) FE model geometry; (B) Boundary conditions applied to simulate falling on the hip; (C) Maximum von Mises stress plot; (D) Internal von Mises stresses.

The total femoral deformation with PMMA and Cortoss implants is similar to normal bone deformation (1.2% maximum difference); however, a stainless steel implant decreases the total deformation by 4.2%. The relatively small deformations resulted from the FE analysis lead to the conclusion that the standard femur model with elastic modulus close to that of bone is too stiff to detect changes due to different implant materials. Therefore, it is important to design implants using FE models derived from CT scans of bones. With such models, patient based FE models can be developed to evaluate if the proposed implant will be beneficial or to optimize the implant mechanical properties.

SUMMARY/CONCLUSIONS

FE analyses of stresses resulted from falling on the hip condition at 1KN, 4KN, and 8KN impact forces were performed. The intact standardized femur model and models with proximal implants made of PMMA, Cortoss, and stainless steel were simulated. The results show that the stresses in the cortical bone decrease when the stiffness of the implant increases and when the diameter of the implant increases. The opposite is true for the trabecular bone: the stresses increase with stiffer implants. Although the standard femur can offer an initial model to study different implantation methods more refined models based on CT scans of osteoporotic patients must be used to select the required material and geometry of an effective implant.

REFERENCES

Currey, J.D. (2002) *Bones: Structure and Mechanics*. Princeton University Press.

Erbe, E.M. et al. (2001) *Eur. Spine J.*, **10**, S147-52.

Heini, P.F. et al. (2003) *49th Ann. Meeting Ortho. Res. Soc.*, Poster 1248.

Viceconti, M. et al. (1996a) *J. Biomech*., **29**, 1241.

Table 2: Maximum von Mises stress (MPa) in subcapital and intertrochanteric regions of the femur due to a 4000 N force on the greater trochanter.

Bone Stress Region	**Normal**	**Implant Size (cm)**	**Implant Material**		
			PMMA	**Cortoss**	**316L**
Subcapital	22.9	**1.0**	22.2	21.7	18.3
		1.5	21.7	20.4	16.8
		2.0	20.4	18.5	14.8
Intertrochanteric	36.9	**1.0**	36.1	36.0	30.4
		1.5	36.1	34.2	25.8
		2.0	35.2	33.3	22.3