STRESS ANALYSIS OF CUSTOMIZED BONE IMPLANTS

M. Rupasinghe, P. Anurathan, R.S. Amarasinghe and S.R. Herath*

Department of Civil Engineering, University of Peradeniya

Introduction

Joint replacement has become a standard procedure in the treatment of patients with crippling diseases such as arthritis and in the treatment of patients who have sustained severe injury due to accidents. Most of the emphasis in total joint replacement research and development has been centered on the knee and hip components (Huiskes et al., 1981). Total elbow replacement, by comparison, has received little attention. Up to date, customized bone implant design is not implemented in Sri Lanka. This study aims to provide an analytical closed form solution for designing of elbow joint implants. Two simple analytical models have been used to study the internal force distribution within the prosthesis and the ulna bone. The effects of stem diameter on the internal stresses developed have been investigated. The model which is in good agreement with the results of a finite element model (FEM) is chosen for further investigation.

Materials and Methods

In order to study the effects of prosthesis geometry on the internal stresses in the implant and the bone, analytical model based analyses together with finite element method based analyses were used (Amarasinghe et al., 2009). External reactions and the muscle force were obtained using a simplified model of the elbow. The two analytical models mentioned above are as follows.

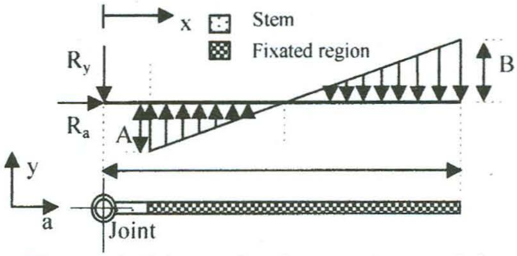

Figure 1. Linear load transfer model

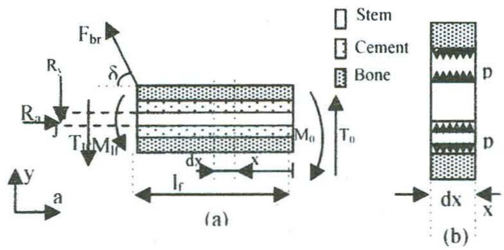

Figure 2. Beams on elastic foundations model (Model 2)

Model 1 (Linear Load Transfer Model): This is a model where the distribution of lateral load transfer along the prosthesis fixation length was assumed to be a planar linear variation as shown in figure 1. The thickness of the cement layer was assumed to be negligible. Ry and Ra are the components of the joint reaction. lpu is the length of the prosthesis and A and B are the values of the linearly transferred load at the ends of the contacted length.

Model 2 (Beams on Elastic Foundation Model): In this model it was assumed that the prosthesis stem and bone behave according to beam theory (Simple Bending Theory) while the cement layer was modeled by a continuous row of linear springs with stiffness properties defined for transverse loading, axial, shear and circumferential shear. Figure 2 gives a schematic representation of this model used for analyzing transverse loading conditions. R_a and R_y are the joint reactions. F_{br} is the force in the Bracialis muscle. M_{lf}, T_{lf}, M_0 and T_0 denote shear force and bending moments at the ends of the contacted length as shown. A similar model was used to analyze axial loading.

The Young's Moduli of bone cement, bone and stem were assumed to be 2, 20, 200 GPa respectively. Poisson's ratio was assumed to be 0.3 for all materials.

Results

Using only circular geometry, following results were obtained. Figure 3 compares the variations in the two analytical models with the FE results (Anurathan et al, 2009) while a comparison of the stress profiles with the diameter of the prosthesis changed is shown in Figure 4 for Model 2.

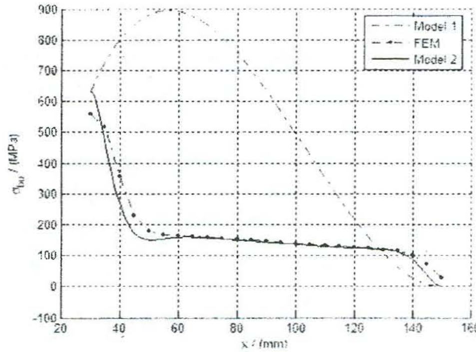

Figure 3. Comparison of top fiber stress FEM results for both models

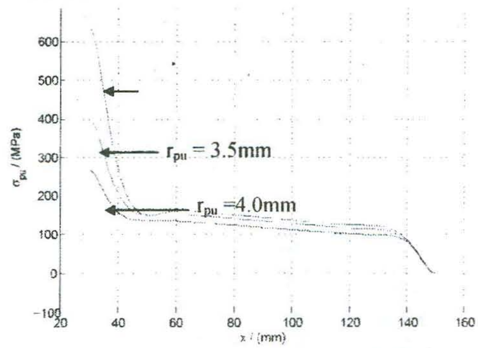

Figure 4. Variations of top fiber stress in prosthesis as the diameter of the prosthesis is changed (Model 2)

Discussion

For the same dimensions of the prosthesis and the bone, it can be observed that the second model is in far better agreement with the results of the FEM. As a result the analytical model 2 was chosen for studying the effects of variation of geometry and dimensions of the prosthesis. From Figure 4, it is clear that the maximum stress concentration occurs at the proximal end of the stem and reduces with the increase in the radius of the cross section. The stresses along the length of the stem are more or less the same.

Conclusion

As the beams on elastic foundation model gives good agreement with the results of the finite element model, it was accepted to be an accurate representation of the elbow join implant. Further analyses with changes in geometry and dimensions were carried out with this model (Amarasinghe et al., 2009).

References

Amarasinghe, R.S., Anurathan, P. and Rupasinghe, R.A.M. (2009), Stress Analysis and Design of Customized Bone Implants, Final Year Undergraduate Project report, Department of Civil Engineering, University of Peradeniya.

Anurathan, P., Rupasinghe, R.A.M., Amarasinghe, R.S. and Herath, S. R. (2009), Effect of Prosthesis Dimensions on the Stresses of an Elbow Joint Prosthesis, Proceedings of the Peradeniya University Research Sessions, PURSE 2009Volume 14, Part I.

Huiskes, R., Janssen, J. D. and Slooff, T. J. (1983), A Detailed Comparison of Experimental and Theoretical Stress-Analyses of a Human Femur, Mechanical Properties of Bone. American Society of Mechanical Engineers, New York, 45, 211-234.