Pre-operative Surgical Simulation of an Acetabular Press Fit: Assembly Strains in Bone

Branislav Jaramaz, Ph.D., Anthony M. DiGioia, III, M.D.
Center for Orthopaedic Research, Shadyside Medical Center, Pittsburgh, PA
and
Center for Medical Robotics and Computer Assisted Surgery,
Robotics Institute, Carnegie Mellon University, Pittsburgh, PA

ABSTRACT

An idealized axisymmetric finite element model of the acetabulum and acetabular implant was analyzed. The assembly strains induced in the bone by the press fit insertion of an oversized cementless acetabular implant were examined. The interaction between implant and bone was modeled as a nonlinear contact-coupled interface to simulate the actual process of press fitting the oversized implant into the prepared acetabular bone bed.

INTRODUCTION

Initial implant stability of the cementless acetabular implant is commonly achieved by press fitting oversized acetabular components. Implicit in this concept is "pre-straining" of the bone which must occur at the time of implantation. Assembly strains are expected to be very different in both magnitude and direction from the strains induced by normal joint forces, and can greatly change the mechanical environment and load transfer patterns.

The mechanical consequences of an interference fit and the development of these assembly strains in bone are critical to our understanding of both the biologic response of bone to mechanical stimuli and the way in which the bone/implant construct may be modeled. The purpose of this study was to determine the effect of the amount of press fit and the size of the implant on the magnitude of resultant assembly strains.

METHODS

Axisymmetric nonlinear, contact-coupled models of a cementless acetabular system were developed to examine the complex interaction between the bone and implant at the time of implantation. The geometry was parametrically described to accommodate for different sizes of the acetabulum and the implant. The resulting finite element mesh was constructed of up to 958 quadrilateral axisymmetric solid elements for the bone and up to 201 solid elements for the implant. The computational analysis was performed on a DECstation 5000/240 using the software package ANSYS 5.0 (Swanson Analysis Systems, Inc., Houston, PA). The finite element code allows for a solution of generally formulated contact between two deformable bodies. This type of formulation permitted the simulation of implanting an acetabular component in a manner that mimics the actual surgery. The implant is initially positioned above the acetabulum and incrementally driven into prepared hemispherical cavity. In its final position, the implant is held in place only by contact with the bone. The forces which tend to eject the implant are equilibrated only by the development of friction at the contact surface.

The material model assumed for bone was elastic-perfectly plastic. Material properties were assigned to bone as: for cortical bone, modulus of elasticity E = 16 GPa and yield stress σ_Y= 175 MPa; for cancellous bone E = 1.0 GPa and σ_Y= 5 MPa. The implant was constructed of a titanium shell with overlying titanium mesh. Geometries were detailed from the Harris-Galante II (HGP II) acetabular implant. The titanium component was modeled as a linear elastic material (E=110 GPA for the shell and E=55 GPa for the mesh). The Poisson‘s ratio was 0.3 for all materials. A value of 0.48 was used for the coefficient of friction at the contact between the bone and the implant. Using this model, the effect of parameters such as the amount of press fit, the implant geometry and the coefficient of friction were examined. The amount of implant oversizing (or nominal press fit) was varied from 1, 2, and 4 millimeters for bone size of 40, 50, and 60 millimeters. Resultant strains in the bone were calculated for each case.

RESULTS AND DISCUSSION

The total equivalent strains at the final position (after the constraints are fully released on the implant) are presented for several cases. These are the strains induced in bone prior to the application of joint loads. Figure 1 shows the results for the same size bone cavity (60 mm) and three different amounts of implant oversizing (1, 2 and 4 mm). Figure 2 shows the effect of the same amount of oversizing (2 mm) on several different sizes of the bone and implant. In all cases the strains predicted by the finite element model were much higher than the yield stress in cancellous bone, and in many cases higher than the yield stress in the cortical shell.

All cases showed similar behavior and could not fully bottom out, i.e., the contact between the implant and the bone could not be established over the entire surface. Instead, at the implant’s lowest position, with the implant still held by applied impact forces, the contact was achieved over approximately 1/3 to 1/2 of its arch length, and as expected, was at the periphery of the acetabulum. After the forces on the implant were released, the contact zone was reduced to 1/5 to 1/4 of the arch due to elastic recoil of the implant/bone system. These results are in very good agreement with recently reported experimental results by McKenzie et al. [2]. They noted that only peripheral contact was achieved in all tested cases and that the contact area does not exceed 20% of all area potential for contact. Although the results of this

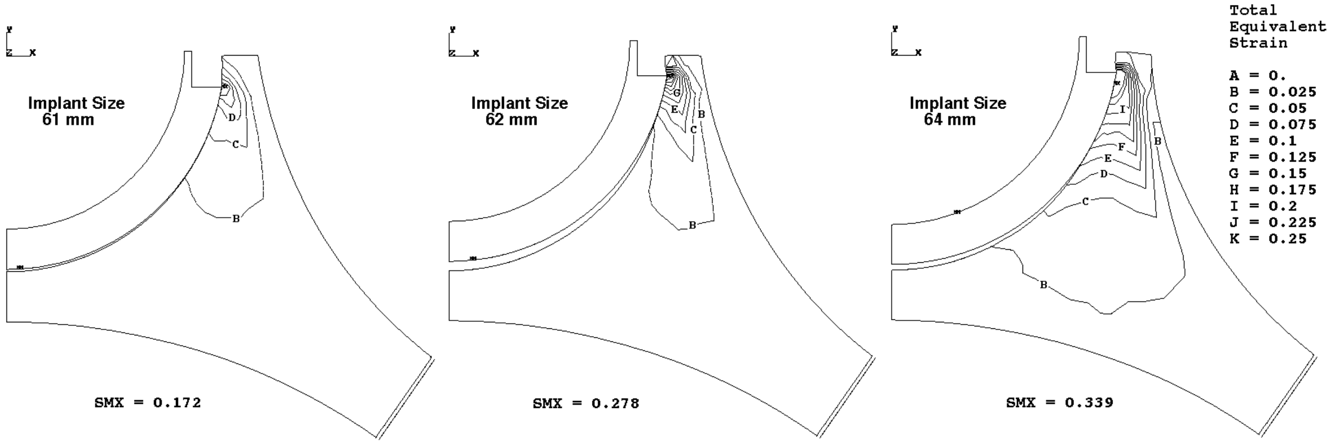

Figure 1 : Total Equivalent Strain for Bone Diameter 60 mm and Implant Oversizing of 1, 2 and 4 mm

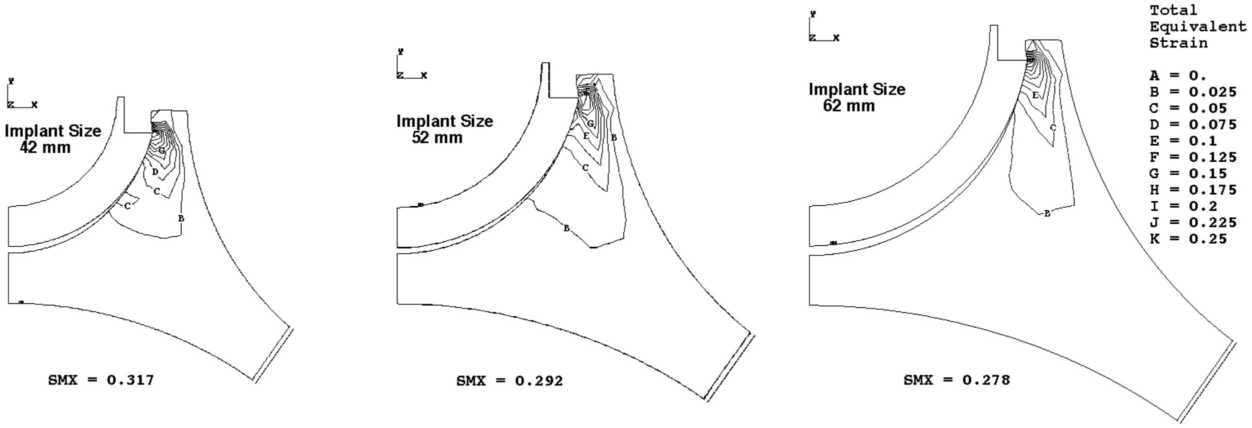

Figure 2 : Total Equivalent Strain for Oversizing of 2 mm and Bone Sizes of 40, 50 and 60 mm

study are qualitatively similar to those previously reported by us [1] for a contact-coupled FE model with linearly elastic material properties, the resulting stresses and strains significantly differ.

The results of this study have several clinical implications. While using a four millimeter nominal implant oversizing may maximize initial stability, as well as maximize peripheral contact thereby reducing potential access of wear debris to the bone/implant interface. However, one effect may be the creation of substantial undesirable gaps between the implant and bone. Furtermore, load transfer is expected to be limited to the periphery of the acetabulum, where the contact is achieved. These factors may ultimately affect the overall implant stability, the ingrowth of bone, the long term load transfer to the bone and bone remodelling process.

Understanding the mechanics of an interference fit and the possible response of bone will permit us to maximize stability while minimizing the potential complications associated with gaps and fractures of bone. This study demonstrated that the surgical procedure of implanting a component into the prepared acetabular bed can be modeled. For many cases, the resultant strains were large enough to compromise the integrity of the cortical bone which could lead to possible fracture. The amount of press-fit and the size of the implant are important factors in the resultant assembly strain distribution and final placement of the component. In most cases, the implant did not bottom out, leaving a gap of various sizes in the polar region between the implant and the bone. In addition, maximizing peripheral press fit is expected to significantly alter the load transfer mechanisms between implant and bone and the resultant stress distribution in bone. These factors in turn are expected to influence the amount and distribution of bone ingrowth and subsequent bone remodeling. Understanding all of these phenomena will help the surgeon to optimize implant stability while preventing bone fractures. In addition, understanding the development of these assembly strains may be useful in the development of precision automated tools to assist the surgeon.

REFERENCES

[1] DiGioia et al. , in 1993 ASME/AIChE/ASCE Summ. Bioeng. Conf., Breckenridge, CO, 1993.

[2] MacKenzie et al., Open Meeting of the Hip Society, San Francisco, CA, 1993.