POLYETHYLENE LINER STRESS IN A SEMI-CONSTRAINED ANKLE PROsthesis

**Miller, M C; *Galik, K; *Conti, S; +*Smolinski, P
**Duquesne University, Pittsburgh, PA. +**University of Pittsburgh, Pittsburgh, PA

Introduction

Large loads on ankle prostheses raise concerns of polyethylene component wear. Commonly used ankle replacement systems employ concentric or nearly concentric articulating surfaces. It is believed that the contact width of the articulating surfaces has an important effect on polyethylene stresses.

A finite element stress analysis was performed for two different talar component designs in the Agility™ ankle system (Depuy, Inc.). The width in the medial-lateral direction of the narrow design varied from 6 mm posterior to 16 mm anterior and the width of the wider design varied from 13.7 mm posterior to 16.2 mm anterior.

Methods

The solid model of the tibia and fibula was created by sectioning and digitizing the distal portion of the lower right leg of a 69-year old female. The solid model of the tibial component of ankle prosthesis was positioned in the solid model of the bones under the direction of a surgeon and the bone removed. This implant system relies on a tibial-fibular syndesmosis fusion and the cortical bone at this location was removed as according to the surgical procedure. No soft tissues in the joints or between the bodies were modeled.

A finite element model of the bone and implant components. The cancellous bone and components were created using ten node tetrahedral elements. The cortical bone in the proximal region was also meshed using ten node tetrahedral elements, but the layer of cortical bone in the distal tibia and fibula was modeled using four node quadrilateral shell elements with a thickness of 1mm. Two 3 mm bone stainless steel screws were represented using beam elements and the bone in the location of the screws was not removed to simplify the meshing.

Isotropic, linear elastic material properties were used for the bone and metal implant components. The polyethylene liner had nonlinear properties [1]. Contact elements were placed between the polyethylene linear and the rigid talar component. All other material interfaces were assumed to be bonded. A total load of five times body weight (body weight = 666 N) was applied to the proximal tibia and fibula with the fibula carrying 7% of the load. The talar component was constrained in all directions and the proximal ends of the tibia and fibula were constrained in the transverse plane. A convergence study showed reducing the element size from 0.25 mm to 0.1 mm gave a change in stress of less than 3%.

Essential results

The finite element analysis was used to compute the stresses in the polyethylene liner. The contact pressure across the polyethylene surface at the apex of the concavity from the medial to lateral side is shown in figure 1. As can be seen from the figure the peak contact pressure at the edges of the implant drops from 36 MPa to 25 MPa with the wider component and the average contact pressure is reduced from 23 MPa to 19 MPa. The von Mises stress has a similar distribution with peaks of 20 MPa in the narrow design and 16 MPa in the wide design at the edges of the contact region. The mean stress decreased from 11 MPa to 9 MPa. It should be noted that the highest von Mises stresses occurred approximately 1 mm below the surface.

Discussion

The wider implant design shows a decrease in both contact pressure and von Mises stress as expected. The computed contact stresses have implications for implant wear. Contact pressures for both implant widths are less than the result of 55 MPa computed for knee replacements [2] and this study used a larger external load. The peak von Mises stresses for both component designs exceeded the yield stress of polyethylene which indicates that some cold flow can be expected.

References