Stress Analysis of Different Rigid Frame Designs within a Flexible Transfemoral Prosthetic Socket

INTRODUCTION

Transfemoral sockets were originally constructed from wood or hard plastic laminates (Radcliffe, 1955). More recently, thermoplastics and acrylic resins with carbon fiber or other woven materials have been used (Ng, 2002). Although these materials achieve suitable load transmission between the residual limb and the prosthesis, their rigidity prevent the sockets from dynamically conforming to changes in residual limb shape and volume during gait (Sanders, 2009).

The ensuing separation (i.e. loss of contact) between the socket and residual limb leads to a loss of negative pressure in suction and vacuum sockets and increase in relative movements (e.g. pistoning) of the residual limb within the socket. A direct, easily achievable solution can be obtained by constructing the socket from a flexible material. However, this solution is constrained by the minimum socket rigidity necessary for effective and stable biomechanical load transfer between the residual limb and the prosthesis.

Our approach to maximizing socket flexibility and maintaining effective load transfer is to construct a single walled, flexible socket, reinforced with a rigid carbon fiber frame (Figure 1). We present results of a finite element (FE) stress analysis evaluating different frame designs.

METHOD

Equipment: Creaform 3-D MegaCapturor digitizer, Novel Pliance system, MTS load system, iPecs unit.

Procedure: A FE model (Figure 1) of a transfemoral sub-ischial prosthetic socket is developed by digital scanning and validated using experimental data (Figure 2). The FE model is formulated to analytically evaluate the performance of three unique frame designs identified with clinical input. The designs are assessed based on the socket-residual limb interface stress magnitude and distribution.

Developing the FE model:

Figure 1: Anterior view of a sub-ischial socket modeled as a simplified two-layered system. A) Scanned 3D vertex data of the socket. B) Solid geometric model. C) FE model is discretized using tetrahedral elements.

RESULTS

Figure 2: Experimental setup to validate FE model. (Left) Loads are applied to a silicon limb model in a custom made socket. (Right) Sensors are inserted between the liner and socket to estimate the normal stress (interface pressure).

DISCUSSION & CONCLUSIONS

The FE results showed a non-uniform interface stress (pressure) distribution that was different for each socket (Figure 3). The sockets differed in the location and extent of cut-outs in the rigid frame. Cut-outs in transfemoral sockets have been used to provide release areas that accommodate displaced tissues (Alley, 2011). The results suggest this approach can be useful to optimize responsive (i.e. flexible) sockets, capable of conforming to a changing residual limb while achieving biomechanical load requirements. Work is ongoing to refine the computational model and to perform more extensive experimental validation. Future work will focus on directly extracting optimal frame designs from the FE analysis.

REFERENCES

Ng, P. Rapid Prototyping Journal, 8(1), 53, 2002.
Alley, DA. et al. JRRD, 48(6), 679-696, 2011.

This work was funded by Department of Defense Award #W81XWH-10-1-0744.

American Academy of Orthotists & Prosthetists
38th Academy Annual Meeting and Scientific Symposium
March 21-24, 2012