A PARAMETRIC FINITE ELEMENT STUDY OF CONSTRAINED ACETABULAR LINERS

Suzanne M. Bouchard[2], Kristofer J. Stewart[1], Douglas R. Pedersen[1], John J. Callaghan[1], Thomas D. Brown[1,2]

[1] Department of Orthopaedics and Rehabilitation, University of Iowa, Iowa City, IA, USA
[2] Department of Biomedical Engineering, University of Iowa, Iowa City, IA, USA
E-mail: Suzanne-bouchard@uiowa.edu   Web: mnypt.obrl.uiowa.edu/OBL_Lab_Website

INTRODUCTION

Dislocation is the second leading cause of failure in total hip replacements, with roughly 2-11% of cases dislocating after the primary surgery. Of revision surgeries for recurrent dislocation, as many as 50% are unsuccessful at preventing further dislocation (Shapiro 2003). Because of the problematic nature of recurrent dislocation, several constrained acetabular cup designs have recently been developed for use as a last-best-effort to maintain stability.

Unfortunately, the clinical outcomes for constrained liners are highly variable. This investigation is a step toward fully characterizing constrained acetabular cups, using a finite element model to explore the design of these devices. Finite element nonlinearity challenges included (1) the interference fit between the femoral head and the cup during intra-operative assembly, (2) appropriate material and kinematic definitions, and (3) cup-to-constraining ring contact. A formulation was developed to vary cup features parametrically. The model was utilized to study both the intra-operative assembly process, and lever-out dislocation.

METHODS

A three-dimensional finite element mesh of a constrained acetabular cup system was developed using PATRAN's pre-processing software. The geometry was tailored to capture the salient design features of constrained acetabular cups from three manufacturers. The baseline model for parametric testing used a 28mm diameter head, with a cup opening radius of 13.75mm. The liner was 8mm thick until just below the equatorial plane, where the thickness was reduced to 6mm to accommodate the metal constraining ring. The articular radius of the cup was 14.5 mm and the inside corner of the lip had a fillet radius of 1.0 mm. Beyond the fillet, the cup receded at a 45º angle (Figure 1, A).

The finite element model contained 31,971 elements with 101,877 degrees of freedom. The femoral component was modeled as a rigid body using triangular facets (Figure 1, B). The metal constraining ring was modeled as titanium with a linear material definition (E = 110 GPa, ν = 0.3), and the UHMWPE cup was given a nonlinear elastic/plastic material definition (Cripton

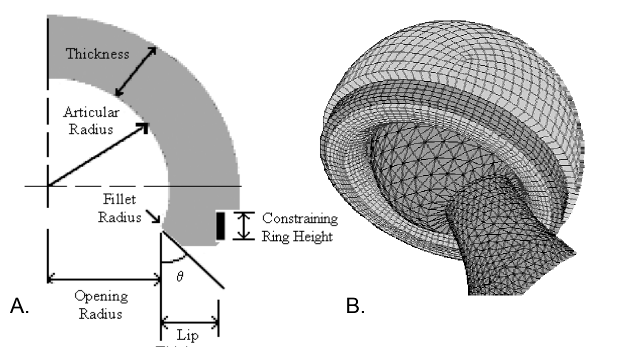

Figure 1: A) Cross-sectional view of the base model. B) Finite element model of the constrained acetabular cup with constraining ring and femoral head and neck, just prior to impingement.

1993). The nodes on the back of the cup above the hemispheric plane were constrained, simulating rigid bony bed support.

The opening radius, lip thickness, and head size were varied parametrically, and simulations of the assembly process and lever-out dislocation were executed for each series using ABAQUS standard finite element code. The key output metric for the assembly process was the force required to insert the head into the cup. For lever-out, the metrics of primary interest were peak resisting moment about the head center, polyethylene Von Mises stress, and rotation angle at dislocation.

RESULTS AND DISCUSSION

As expected, the opening radius had a significant effect on the force required to push the femoral head into the cup. For a series of five cups with increasing opening radii from 13.6 to 13.9 mm, the reaction force ranged linearly from 39.3 to 545.5 N ($R^2 = 0.966$). The moment about the center of the head as the femoral component rotated out of the cup decreased monotonically (Figure 2).

For a series of six liners with varied lip thickness from 4.5 to 7 mm, the relationship with assembly force also increased linearly with a range of 222.8 to 282.3 N ($R^2=0.97$). The maximum moment resisting rotation of the head was much less affected in this series, but decreased from 4.79 N-m to 4.63 N-m as the lip thickness increased.

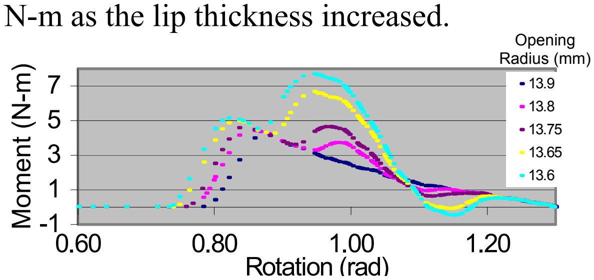

Figure 2: Moment about head center as the head rotated out of the cup

Unlike the relationships seen for assembly force in the previous two series, there was no linear correlation between head size and assembly force (Table 1). As expected, the range of motion increased with head size (Figure 3), although the peak resisting moment did not follow this trend.

Table 1: Effect of head size on assembly force

Head Size (mm)	22	26	28	32	36	40
Force (N)	469	266	240	313	341	250

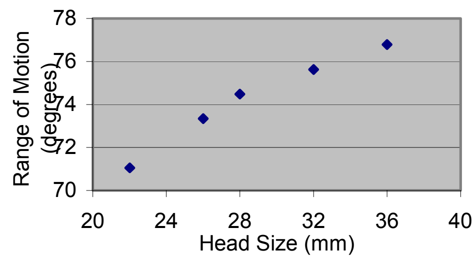

Figure 3: Angle of rotation at dislocation

CONCLUSION

When the lip thickness is reduced the constraining ring plays a greater role in the liner's resisting moment, but the stress in the UHMWPE at the motion limit is also increased. Increasing head size alone increases the motion, but not the peak resisting moment. Because the interactions between design parameters can be complex, finite element modeling is very beneficial for exploring design alternatives.

REFERENCES

Shapiro, G.S., et al. (2003). *The Use of a Constrained Acetabular Component for Recurrent Dislocation. J. Arthroplasty*, **18**, 250-258.

Cripton, P.A., (1993) Thesis. Queen's University.

ACKNOWLEDGEMENTS

Funding source: DePuy Orthopaedics Inc.