VALIDATING A COMPUTATIONAL MODEL OF REVERSE SHOULDER ARTHROPLASTY

Vijay N. Permeswaran, Carolyn M. Hettrich, Jessica E. Goetz, Donald D. Anderson

The University of Iowa, Iowa City, IA, USA
email: vijay-permeswaran@uiowa.edu, web: http://tinyurl.com/UIOBL

INTRODUCTION

Reverse shoulder arthroplasty (RSA) is indicated for treating end-stage arthritis when the rotator cuff is compromised. In RSA, the native anatomy of the ball-in-socket joint is reversed by implanting a cup onto the humerus and a sphere onto the glenoid (Figure 1). A common complication is impingement between the humeral cup and inferior scapular ridge, which causes scapular bone erosion (notching). In order to prevent notching, surgeons lateralize the center of rotation of the glenosphere. We previously presented a finite element (FE) analysis of the effect of lateralization on range of motion and stress at the impingement site [1]. This study aimed to validate that prior FE analysis by measuring the contact stress at the impingement site in a cadaveric model.

Figure 1: The image on the left shows a native shoulder, while that to the right shows a shoulder after RSA.

METHODS

A cadaveric shoulder was dissected and prepared using the bony increased offset (BIO) technique [2]. The proximal humerus was transected 5 cm distal to the deltoid insertion. The medial edge of the scapula and distal end of the humerus were both potted in PMMA. A Tornier Reversed Aequalis RSA system (Tornier, Amsterdam, Netherlands) was implanted. Metal spacers replicating BIO offsets were used to study lateralizations of 0, 2.5, 5, 7.5, and 10 mm. A custom loading frame was constructed to secure the scapula. The humerus was held in place by a cable representing the deltoid muscle. This cable was clipped onto an eyehook screwed into the distal humerus PMMA block, routed through a pulley to replicate the wrapping of the deltoid, and attached to the load frame by a turnbuckle, which facilitated adjustment of tension in the deltoid cable.

CT scans were obtained for the cadaveric specimen before and after RSA implantation. The scapula, humerus, and implant components were segmented using Seg3D software (CIBC, Salt Lake City, UT). The FE model was generated from these surfaces.

A Tekscan ankle sensor (model #5033; Tekscan, South Boston, MA) was selected to measure the impingement contact stress because of its high spatial resolution (sensel resolution of 0.834 mm x 0.834 mm). A hole was punched in the sensor in order to secure it to the post of the glenosphere. The sensor was then pinched between the glenosphere and metal spacer. A portion of the sensor around the punched hole was trimmed to eliminate major wrinkling (Figure 2). During testing, a 5 lb. load

Figure 2: Tekscan sensor alignment and positioning. The lateral end of the tekscan sensor was modified and pinched between the glenosphere and lateralization spacer.
was hung from the humeral PMMA block. The deltoid cable was tensioned to prevent subluxation. In addition, a Qualisys Oqus motion tracking system (Qualisys AB, Gothenburg, Sweden) was used to record the position of the scapula and humerus at impingement.

The FE analysis was done using Abaqus/Explicit (Dassault Systèmes, Vélizy-Villacoublay, France). The global model included deformable glenoid and BIO bone, a rigid glenosphere, a deformable polyethylene cup, a rigid humeral component, and a rigid humerus. In addition, a series of cable and pulley elements were used to model the deltoid muscle, and spring elements were used to represent the joint capsule. The humerus was adducted until impingement and then fixed in position. Due to the highly focal nature of the contact patch, a high-resolution submodel of the impingement site was created. Boundary node displacements were applied using the submodel option. A custom Matlab (MathWorks, Natick, MA) script was used to analyze and compare the Tekscan results to the results of the FE analysis. Contact stress, area, and recovered load values were analyzed as well as location and contact patch morphology.

RESULTS AND DISCUSSION

Only results for the 7.5 mm implantation are here reported. The FE-computed maximum contact stress was 30.9 MPa, while the maximum contact stress for the Tekscan sensor was 21.2 MPa. The FE-computed contact area was 1.05 mm². The contact area of the two primary Tekscan sensels was 1.4 mm². The load transferred across the FE-computed contact patch was 39.9 N. The Tekscan sensor provided a loading of 42.3 N.

Figure 3 shows how the Tekscan sensor and FE model locations and areas correspond. The FE analysis reported a small contact patch indicative of the edge contact that occurred between the humeral cup and inferior ridge of the glenoid. The Tekscan sensor reported two sensels with contact significantly higher than all surrounding sensels (21 and 19 MPa vs. 2-5 MPa). When overlaid, the FE-computed contact patch aligned well with the two highly loaded sensels. This result provides a potential explanation for why the two sensels measured a smaller contact stress than that computed in the FE analysis. In addition, the FE model would be expected to create smaller contact area than the Tekscan due to the difference in spatial resolution. Finally, the Tekscan sensor can wrinkle around load application sites, yielding artifactually low contact stress readings. The similarity in contact load calculated is an indication that difference in contact area is likely the cause for dissimilar maximum contact stress.

CONCLUSIONS

In conclusion, a finite element analysis of the mechanical consequences of lateralization in RSA was successfully validated against cadaveric measurement of contact stress. This FE model can be used to computationally assess novel implantation procedures prior to clinical application.

REFERENCES

1. Permeswaran et al. 37th ASB Annual Meeting, 201, Omaha, NE, USA, 2013

ACKNOWLEDGEMENTS

This work was partially funded by a New Investigator Grant from the Orthopaedic Research and Education Foundation.