An Explicit Finite Element Model of Total Elbow Replacement Contact Mechanics

Danial Sharifikia, Ryan Willing
State University of New York at Binghamton, Binghamton, NY, USA.

Disclosures:

Introduction:

Total elbow replacement (TER) is a common treatment for end-stage elbow joint arthritis or fracture. Although TER results in joint motions similar to the natural joint, bearing wear, excessive deformations or early fracture might affect the final outcome of the replacement. Computational models have been widely used to evaluate the behavior of different implant designs under physiological loading. However, as TER is less common than other arthroplasty techniques, less research has been focused on the evaluation of the outcomes of TER, failure mechanisms and development of optimal designs. The current study aims to develop Finite Element computational models of TER implants in order to simulate the effects of different loading patterns on bushing contact stresses and to predict possible failure mechanisms.

Methods:

A geometrical model of a TER assembly has been developed based on measurements from a Coonrad-Morrey TER implant (Zimmer, Inc., Warsaw, IN) (small, left), one of the most commonly used implant designs for TER. The assembly was discretized using a combination of hexahedral elements for the bushing components and areas where contact was simulated, and tetrahedral elements for the remaining surfaces (Fig. 1). Ultra high molecular weight polyethylene (UHMWPE) material properties were assigned to the humeral and ulnar bushings (E=690 MPa, v = 0.46). A penalty contact formulation was defined between all of the surfaces of the model to take into account every possible interaction between different implant components in vivo. A coefficient of friction of 0.04 was used to consider the effects of frictional forces in tangential contact directions. The loading applied to the model includes a flexion-extension motion with an amplitude of 110 degrees, a joint force reaction with variable magnitude and direction and a time varying varus-valgus (VV) moment with maximum magnitude of 5 Nm; loading which has been previously used in other work and has been proposed as a standard for pre-clinical testing of TER. An explicit FEM solver was used in order to account for dynamic motions of the implant, inertial effects and large displacements of the components during a loading cycle. All model development and analyses were performed using the commercially available FE analysis software ABAQUS (Dassault Systèmes, Vélizy-Villacoublay, France). The model predicted contact pressures were validated through comparison with Hertz contact formulations. Model results were compared directly with corresponding experimental data. Experimental wear tests were performed on the abovementioned implants using a VIVO (AMTI, Watertown, MA) six degree-of-freedom (DOF) joint motion simulator apparatus. The worn TER bushings were scanned after the test using micro computed tomography (µCT) imaging techniques, and reconstructed as 3D models. Comparisons were made between the possible sites of failure based on the numerical simulation and the experimental results.

Results:

Contact pressure distributions on the humeral and ulnar bushings are demonstrated in Figure 2 for a single instance during the dynamic loading cycle, as well as µCT-based models of the humeral and ulnar bushings. Arrows indicate regions of excessive deformation or creep for comparison purposes. Contact pressures on the humeral and ulnar bushings reached magnitudes as high as 77 and 152 MPa respectively, during a loading cycle. As a result of the varus-valgus moment, the ulnar component of the implant rotates and comes into contact with humeral bushing generating high localized contact pressures. Plastic deformation and creep is evident in experimental results close to the regions with high contact pressures in the numerical simulation.

Discussion:

The results represent high localized contact pressures on the bushings which can be a reason for poor outcomes due to wear or early fracture of the TER implants. Numerical results demonstrate acceptable correlations with experimental data based on the location of deformation and creep on bushings after experimental tests. The numerical model developed can be further used to simulate the effects of different loading patterns on the implant (eg, chair-rise) and to evaluate the outcome of optimized TER designs. Furthermore, by the manipulation of the data provided by FE simulation and the application of a wear criteria, the volume of material removed due to wear can be evaluated which leads to better understanding of critical sites of failure on the implant.

Significance:

Numerical simulation of the behavior of TER implants provides essential data on the location and distribution of contact stresses and wear mechanisms. The current study develops an explicit finite element model to evaluate implant behavior and contact stresses due to loading.