Elasto-Plastic Computational Modelling of Damage Mechanisms in Total Elbow Replacements

Danial Sharifi Kia, Ryan Willing
Department of Mechanical Engineering, State University of New York at Binghamton
Binghamton, NY, USA

As a treatment for end-stage elbow joint arthritis, total elbow replacement (TER) results in joint motions similar to the intact joint; however, bearing wear, excessive deformations and/or early fracture may necessitate early revision of failed implant components. Compared to hips, knees and shoulders, very little research has been focused on the evaluation of the outcomes of TER, possible failure mechanisms and the development of optimal designs. The current study aims to develop computational models of TER implants in order to analyze implant behaviour; considering contact stresses, plastic deformations and damage progression.

A geometrical model of a TER assembly was developed based on measurements from a Coonrad-Morrey TER implant (Zimmer, Inc., Warsaw, IN). Ultra high molecular weight polyethylene (UHMWPE) nonlinear elasto-plastic material properties were assigned to the humeral and ulnar bushings. A frictional penalty contact formulation with a coefficient of friction of 0.04 was defined between all of the surfaces of the model to take into account every possible interaction between different implant components in vivo. The loading scenario applied to the model includes a flexion-extension motion, a joint force reaction with variable magnitude and direction and a time varying varus-valgus (VV) moment with a maximum magnitude of 13 N.m, simulating a chair-rise scenario as an extreme loading condition. An explicit dynamic finite element solver was used (ABAQUS Explicit, Dassault Systèmes, Vélizy-Villacoublay, France), due to improved capabilities when performing large deformation analyses. 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 (6-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 based on the sites of damage and deformed geometries between the numerical results and experimental test data. In addition to that, parametric geometrical models were developed using worn geometry of the retrievals in order to account for primary wear and deformations while simulating long-term contact stress and secondary damage progression on the bushings (Fig. 1).

Contact pressure distributions on the humeral and ulnar bushings correlate with the sites of damage as represented by the μCT data and gross observation of clinical retrievals. Furthermore, deformation patterns and kinematics of the components are in good agreement with the experimental results (Fig.2). Excessive plastic deformations are evident in both the numerical and the experimental results close to the regions with high contact pressures. Simulating parametric initially-worn geometries results in the formation of secondary damage zones, as well as redistribution of contact stresses and contact locations (Fig. 3).

The results demonstrate UHMWPE bushing damage due to different loading protocols. Numerical results demonstrate strong agreement with experimental data based on the location of deformation and creep on bushings and exhibit promising capabilities for predicting the damage and failure mechanisms of TER implants.

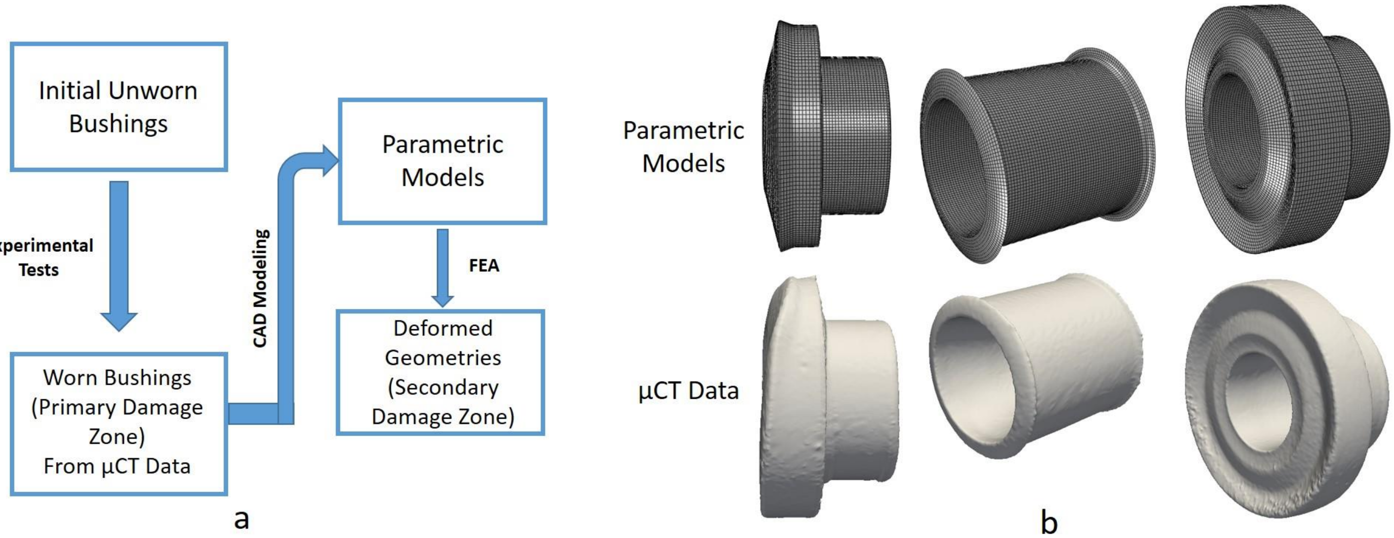

Figure 1: a) Parametric modelling procedure in order to analyze bushing damage progression b)Parametric modeling of worn bushing geometries based on μCT data

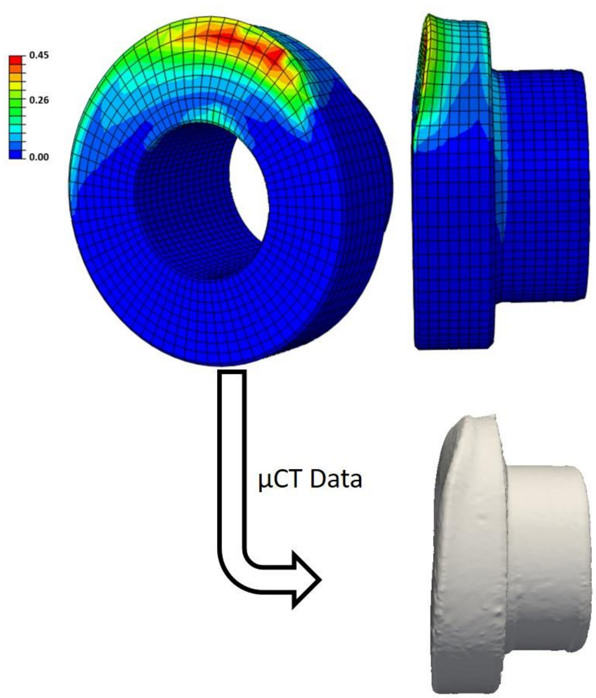

Figure 2: Plastic strain distribution on the lateral humeral bushing compared to μCT data from experimental tests

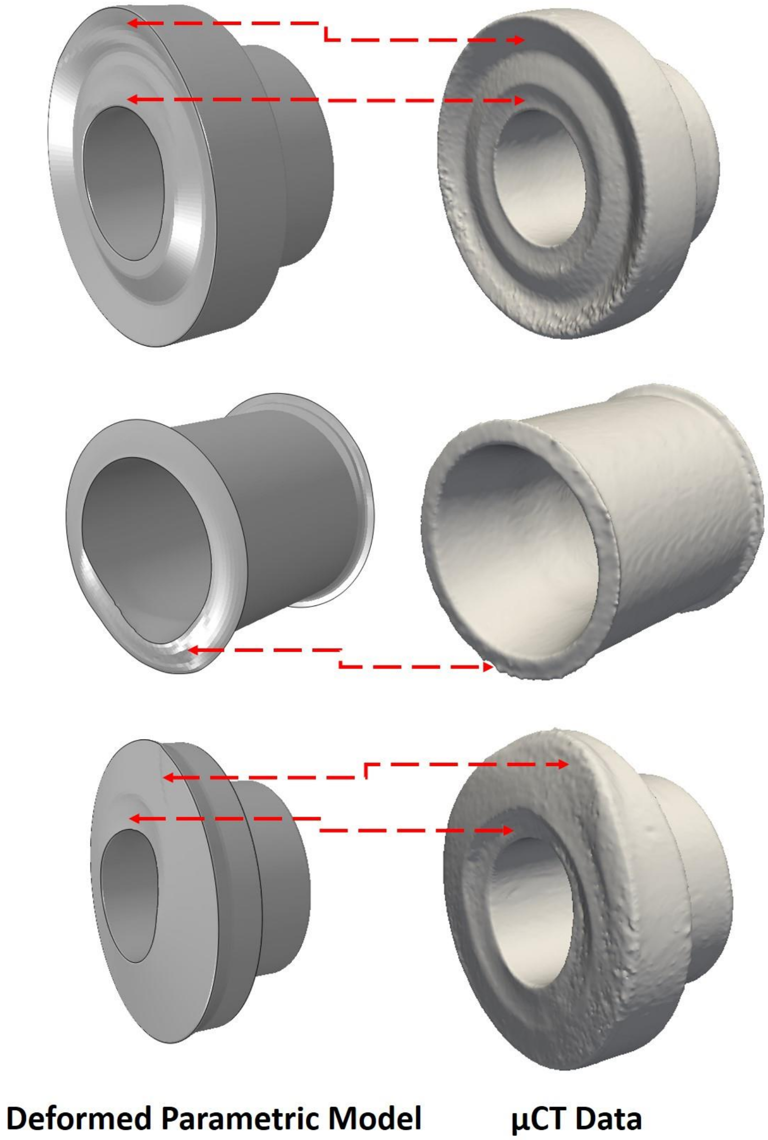

Figure 3: Simulation results demonstrating damage progression and formation of secondary damage zones