Introduction: In this study, we developed a coupled structural and kinematic finite element model of a total knee replacement to evaluate the polyethylene stresses and tibial motion during loading in a Stanmore knee simulator that operates under load-control. We chose to model the loading of a knee simulator for several reasons: 1) in contrast to in vivo knee kinematics, the kinematics of the knee simulator were easily defined based on the machine inputs; 2) as the duty cycle used in this particular knee simulator has been proposed as a basis for the ISO test standard, it is important for us to gain an understanding of associated polyethylene stresses under load control to compare them with in vivo predictions; and 3) it will ultimately allow us to run repeatable, in vitro simulations, and correlate mechanical parameters (e.g., contact stress, von Mises stress) with observed polyethylene damage modes. Recent analyses of total knee replacements have decoupled the joint kinematics from the stress analyses, treating them as separate problems [1]. In the present study, the joint kinematics and stress analyses were coupled effectively in load-control throughout the analysis of the gait cycle. The long-term goal of this research is to validate a numerical knee simulator to develop an improved understanding of the mechanism of polyethylene wear and surface damage in total knee replacements.

Methods: Three-dimensional finite element models were constructed from CAD surface files of a Size 7 cruciate retaining total knee replacement (Scorpio, Howmedica Osteonics, Allendale, NJ). Meshes of the femoral and the tibial components were created using TrueGrid (XYZ Scientific Software, Livermore, CA). The femoral component consisted of 5,320 shell elements and the tibial insert with base plate consisted of 24,960 solid elements (Fig 1). The femoral component and the tibial base plate were modeled as rigid bodies. The insert was modeled as an elasto-plastic material, having a modulus of 1016 MPa, Poisson's ratio of 0.46, and yield behavior defined from uniaxial compression testing [2]. Pairs of nonlinear springs were attached to the posterior edge of the tibial base plate to simulate the posteriorly located spring buffers of the knee simulator. Loading and boundary conditions were based on the ISO/DIS duty cycle, as implemented in the Stanmore knee simulator (Fig 2). The center of gravity of the femur was constrained in all directions with a flexion angle defined to rotate about the center of rotation of the posterior condyle (Figs 1,2). The base plate was free to translate in the anterior-posterior (AP) direction in response to the applied axial and AP loads. The simulation was carried out over the first 0.6 s of the ISO/DIS duty cycle, corresponding to the stance phase of gait. The coupled kinematic/stress analysis was simplified by neglecting the axial torque. Tibio-femoral contact was defined with a coefficient of friction of 0.085. The contact stress, von Mises stress) with observed polyethylene damage modes. Recent analyses of total knee replacements have decoupled the joint kinematics from the stress analyses, treating them as separate problems [1]. In the present study, the joint kinematics and stress analyses were coupled effectively in load-control throughout the analysis of the gait cycle. The long-term goal of this research is to validate a numerical knee simulator to develop an improved understanding of the mechanism of polyethylene wear and surface damage in total knee replacements.

Results: Maximum stresses and strains in the polyethylene were observed at ~50% of the ISO/DIS duty cycle. The maximum von Mises stress was 22.4 MPa, the maximum contact stress was 27.2 MPa and the maximum von Mises strain was 0.029. The von Mises stress distribution showed elevated stresses in the central portion of the contact region, with a small area of higher stress located towards the inner spine of the polyethylene insert. The shape of the contact stress patch (Fig 3) extended in an anterior-posterior strip across each of the medial-lateral (ML) radii of the tibial insert. The pattern of von Mises strain also extended as a strip in the AP direction and extending across the ML radii. The patterns of von Mises stress, contact stress and von Mises strain were nearly symmetric between the right and left tibial radii for the loading conditions applied in this analysis. The base plate translated from 1 mm in the anterior direction to 3 mm in the posterior direction. An initial posterior displacement of the base plate was observed during heel strike. After the initial “seating” of the component, the component moved anterior with the anterior force, then posterior with the posterior force.

Discussion: In this study, we used a coupled structural and kinematic finite element model of the knee to evaluate the stresses in the tibial component during loading in a Stanmore knee simulator. Knee replacements of the same design have previously been tested to 3 million cycles in the Stanmore simulator and show burnishing and permanent deformation along the central regions of the condyles, in the areas we observed the maximum contact stresses. These findings are consistent with the magnitudes of maximum stress and strain which are sufficient to induce permanent polyethylene deformation [2], but are below the polymer yield point, which is associated with the onset of crystalline plasticity. Additionally, during this physical testing in the knee simulator, the base plate was translated from 1 mm anteriorly to 2 mm posteriorly, which is in good agreement with our model predictions. It is likely that these results are design specific and will vary with the inclusion of a torsional load. Although results thus far have compared favorably with the physical simulator output, further analyses including torsion will be necessary for a comprehensive validation of the numerical knee simulator.

Acknowledgements: Supported by a grant from Howmedica Osteonics.


Figure 1. Femoral and tibial FE meshes with applied boundary conditions.

Figure 2. Applied force and displacement.

Figure 3. Maximum contact stress solution.