Validation of a Finite Element Contact Model for the Use in Total Knee Arthroplasty

F. Amirouche\textsuperscript{1}, R. Giachetti\textsuperscript{1}, L. Aram\textsuperscript{1}, M. Gonzalez\textsuperscript{2}

\textsuperscript{1}University of Illinois at Chicago, Department of Mechanical Engineering, Chicago/IL
\textsuperscript{2}University of Illinois at Chicago, Department of Orthopaedic Surgery, Chicago/IL

Introduction:

The objective of this research is to identify the dynamic contact pressures within the prosthetic knee experimentally and develop a finite element model which then can be validated. Both the experiment and the model are unique and provide pressure data that is essential to the balancing of the knee during TKA. Experimental data was collected on a cadaver study by means of a distributed array of pressure sensors fixed to the condyles. Each sensor measures, in real time during flexion and extension the contact pressures over a 2 mm diameter area. A finite element contact model of a prosthetic joint was developed and validated using the experimental data. The time history of the contact stress on the knee components were studied in relation to understanding the fitting of prosthesis in knee replacements.

A finite element model was created using ANSYS software to investigate the force distribution of the tibial component. The model was then validated to ensure accuracy. The validated model creates a venue for further stress analysis as a function of alignment scenarios, and is the basis for a fitting tool that can gage the alignment of a prosthesis during the TKA procedure.

Measuring tools currently used during joint replacements often result in tolerances that can exceed 5 degrees of ideal alignment of the knee joint. Rotational alignment, particularly of the tibial component can be difficult to quantify. Increases in stresses due to poor component balancing or alignment can create excessive polyethylene wear. This can lead to premature polyethylene component failure. Polyethylene particulate causes osteolysis, mediated though a macrophage/cytokine cascade. The ensuing bone loss can cause can cause tibial component failure from degradation of the bone cement interface and from loss of the mechanical support of the tibial tray.

METHODS:

The creation of the finite element model involved three stages. Stage 1: Scanning the actual prosthesis with a laser micrometer. Stage 2: The scanned IGES file was imported to Autocad for cleaning. Using Autocad, the surface model was deconstructed into an Ansys friendly wireframe model. Stage 3 (Creation of the solid model): The model was exported from Autocad into the Ansys finite element software for creation of the solid model and execution of the analysis. Using approximately 8,000 3-D elements a model was created. The model used the accepted properties for UHMWPE for the tibial component and the femoral component was modeled as a rigid body. Each was coated in contact elements that use a Gauss point algorithm to determine the contact properties.
Because this was the perfect scenario, a symmetric model could be used so only one half of the femoral component was used to load the model, thus reducing solution time. Loading consisted of the 13 pounds that was used in the experiment to simulate relaxed muscle forces in the muscles of the upper leg. This was distributed over the bottom of the tibial model and the femoral surface was constrained to zero displacement.

RESULTS & DISCUSSION:

Table 1. Contact Pressure

<table>
<thead>
<tr>
<th>Flexion Angle</th>
<th>Experiment (KPa)</th>
<th>FEA (KPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Anterior</td>
<td>Center</td>
</tr>
<tr>
<td>0 °</td>
<td>16</td>
<td>1</td>
</tr>
<tr>
<td>20 °</td>
<td>10</td>
<td>2</td>
</tr>
<tr>
<td>40 °</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>60 °</td>
<td>1</td>
<td>3</td>
</tr>
</tbody>
</table>

To compare the model to the experiment, values of contact pressure were recorded in the areas which correspond to the sensor locations on the prosthesis. At zero degrees flexion angle, the FEM reads 21-psi at the center condyle while the experiment recorded 16-psi at the anterior of the condyle. At twenty degrees the experimental and FE results were the same with 10-psi at the anterior of the condyle. At forty degrees the FE read 13-psi while the experiment recorded 2- and 3-psi. And at sixty degrees the FE read 15-psi while the experiment recorded 25-psi.

The percent between the finite element model and the experimentally recorded values vary from 0% to 333% for different angles. If you take the sum of the three sensors and data points the percent error varies from 17% to 117%. This error can be explained by two factors. First, with the current surgical instrumentation it is very difficult to achieve a perfect fit. The data used in this study is an average of three TKAs. Second, the current finite element model does not account for the forces contributed by the ligaments.
and capsules. A more elaborate FEM is currently being developed that includes the ligaments and capsule as quadratic springs.

**Acknowledgements:**
The authors greatly appreciate the support of the J&J Depuy Corporation.