Hip to Simulation

Evaluation of designs for a hip replacement prosthesis overcomes physical and scientific limitations.

By Joel Thakker, Integrated Design and Analysis Consultants, U.K.

Hip replacement surgery involves replacing the damaged or diseased ball-and-socket joint configuration with artificial parts. During surgery, a cup or hip socket — a dome-shaped shell/liner — is implanted into the acetabulum portion of the pelvic girdle after the bone has been hollowed out using a grater. The thigh, or femoral, portion of the hip replacement prosthesis is composed of a ball, which acts like a bearing where it fits into the cup and is attached to a stem that further attaches to the femur. The Duraloc® uncemented acetabular hip socket, a replacement cup developed by DePuy Orthopaedics, Inc., in the U.K., uses an interference fit to hold the socket in place in the hip bone. To assist DePuy in the design of the Duraloc product, Integrated Design and Analysis Consultants (IDAC) used ANSYS Mechanical software to develop parametric models that are used to establish both the necessary implantation and disassembly forces for variations of the replacement joint.

The Duraloc® uncemented acetabular hip socket is made from titanium and has a porous coated shell.

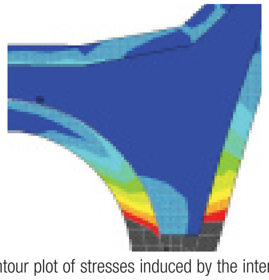
IDAC performed a two-dimensional analysis on the cup assembly in order to model the force required to remove the socket axially. A three-dimensional model was used to analyze rotational removal of the joint, since a two-dimensional case would not represent the behavior fully. The ANSYS Mechanical simulation used nonlinear contact elements in the prosthetic hip socket and accounted for friction between the cup and bone. In all analyses, the implant cup was modeled in titanium while the bone was treated as an anisotropic material.

For both analyses, IDAC created parametric models in order to evaluate different bone and implant cup geometries, material properties and boundary conditions. The assembly conditions involved inserting the cup into the bone to overcome interference, allowing the frictional effects to hold the cup in place, and subsequently removing, either axially or rotationally, the cup from the bone to establish disassembly loads.

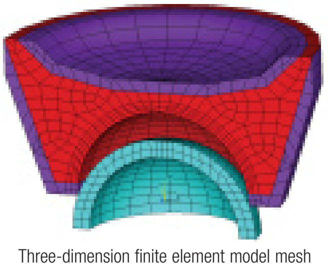
This form of modeling allows DePuy to evaluate different configurations of implant design numerically rather than by physical testing, which is time-consuming and expensive in comparison. Physical testing is limited as real bone materials are not highly available. Some synthetic and naturally occurring materials can be used, but their material properties do not precisely match that of human bone materials. Numerical modeling allows DePuy to view detailed stress and deflection distribution plots and load versus time history plots that cannot be created easily from physical tests. Comparisons between the results obtained through simulation and those obtained from previous testing reveal a close correlation.

X-ray of a hip showing a prosthesis, including the socket, ball and stem. Image courtesy DePuy Orthopaedics, Inc.

As a result of this study, DePuy has used this type of design evaluation in other orthopedic implant products, including artificial knee joints. ■



Contour plot of stresses induced by the interference fit between the prosthesis and the bone; the areas colored in grey illustrate the region of the bone that could be expected to yield during the assembly process.



Three-dimension finite element model mesh of bone and prosthesis

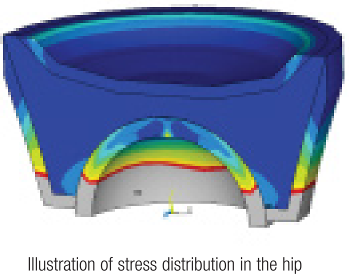


Illustration of stress distribution in the hip joint assembly after the prosthesis has been pressed into place