Design of a Femoral Canal Sizing Device Using ABAQUS

An Undergraduate Senior Design Project at Cedarville University

Senior Design Team Students:
Jason Auyer, John Simmons, Matt Spena and Mary Todd Hofner

Advisor: Timothy L. Norman
Professor of Mechanical and Biomedical Engineering
Cedarville University

***Abstract:* ABAQUS was used to model the push-out load between a hip prosthesis stem and bone with bone-stem contact area and interference, stem-bone friction coefficient and viscoelastic material properties as variables. The push-out loads were used with a device that maps out the geometry of the femoral intramedullary canal to aid a surgeon with optimal prosthesis size selection in real time during hip replacement surgery.**

Introduction

Total hip replacement is a common surgery to correct an arthritic or severely damaged hip joint. There are two major forms of hip replacement surgery: cemented and cementless. The cementless prosthesis depends on stem-bone contact area and "press-fit" for initial stability, and bone growth into the porous stem for long term stability [1-3]. Failure to achieve long term stability requires subsequent prosthesis revision surgery. Instability may be due to a lack of initial or short term stability caused by inadequate initial "press-fit." Inadequate contact area and interference may be attributed to the difficulty in sizing both the reamer, used to create the hole in the femur, and the prosthetic shaft that is inserted into the reamed bone. Often the reamed intramedullary canal has an irregular oval-shaped cross-section that may be different than the shape of the stem if the bone is not completely reamed. If the geometries differ more than the planned interference then contact area is reduced and press-fit will be compromised. Knowledge of the shape of the intramedullary canal and stem prior to reaming and stem installation could aid surgeons in selecting proper reamer and stem sizes.

Currently the most detailed method of constructing a 3D model of the reamed bone is using computer aided tomography, but this is often unavailable in the operating room or may be considered excessive and unnecessary by the surgeon. At this time, the most common means of estimating the size and shape of the intramedullary canal is by x-ray. The x-ray provides a 2D view of an irregular 3D space from which the surgeon estimates the best fit for the prosthetic shaft. As a result, the surgeon may resort to trial-fitting different reamers and shafts to obtain the best feel. If the shaft is too large in comparison to the canal, the bone will fracture, but if the shaft is too small there may be inadequate contact area or press-fit and the implant may be unstable. The difficulty in determining the proper press-fit may be one of the major causes of implant instability.

The goal of the Cedarville University Undergraduate Design Team was to provide the orthopaedic surgeon with real-time 3D information about the shape and size of the femoral canal that can be used in prosthesis selection. Figure 1 depicts the overall design concept. Once the femoral canal is mapped, it is imaged (Figure 2) and contact area is calculated for various reamer and stem sizes. Stem selection is done in real time by comparing FEA predicted stem push-out loads. Figure 2 and Table 1 show examples of the data available to the surgeon during hip replacement surgery. The undergraduate engineering students at Cedarville University designed the femoral canal sizing device (Oculus) used to collect femoral canal measurements and performed the analysis using ABAQUS over the course of a two semester capstone senior design project. Jason Auyer, John Simmons, Matt Spena and Mary Todd Hoffner were advised by Dr. Tim Norman on the project. Mary Hoffner performed the finite element analysis.

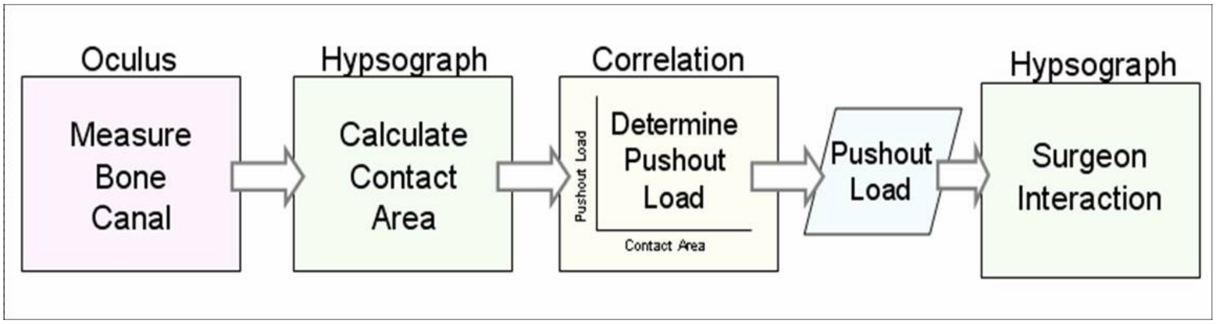
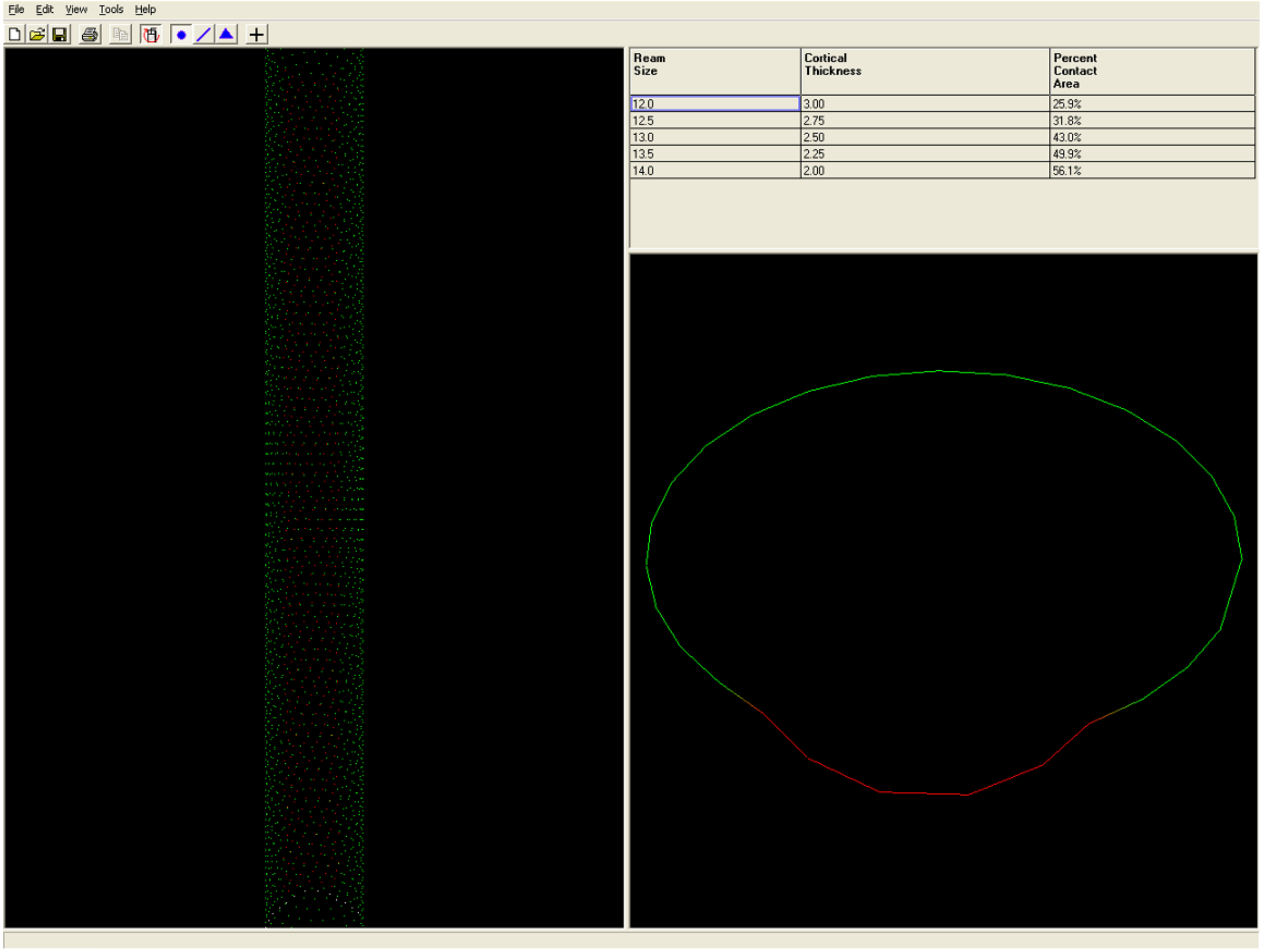


Figure 1: Flowchart of real-time information provided to the surgeon.

Table 1: Table of data displayed in the OR. The push-out load was calculated by ABAQUS.

Ream Size (mm)	Cortical Thickness (mm)	% Contact Area	Push-out Load (N)
12	3	25.9	436.8
12.5	2.75	31.8	501.5
13	2.5	43	602.3
13.5	2.25	49.9	654.2
14	2	56.1	691.3



Ream Size	Cortical Thickness	Percent Contact Area
12.0	3.00	25.9%
12.5	2.75	31.8%
13.0	2.50	43.0%
13.5	2.25	49.9%
14.0	2.00	56.1%

(a) (b)

Figure 2: Illustrations of the (a) longitudinal and (b) transverse sections of the femoral canal as measured by FCSA. Contact area is indicated in green.

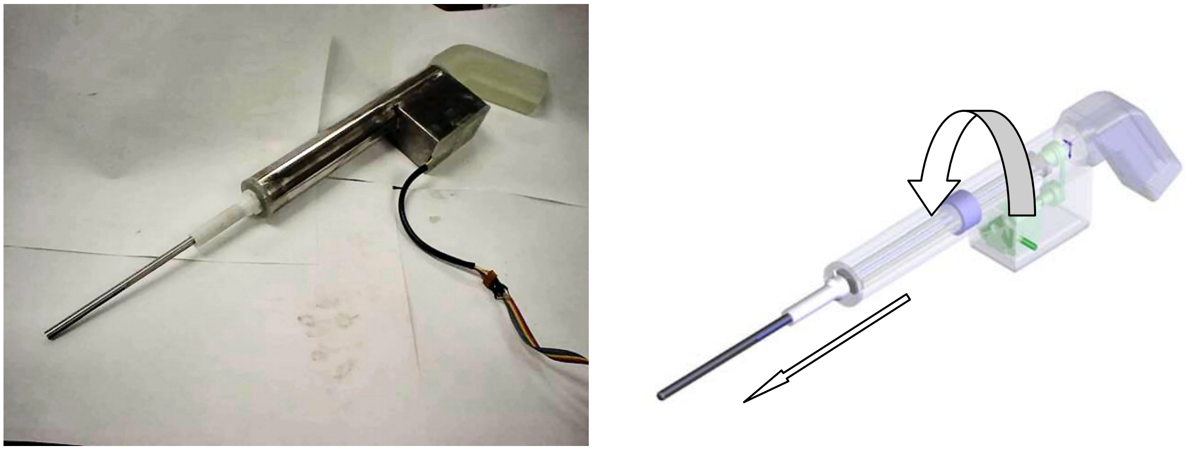


Figure 3: Femoral Canal Sizing Device (Oculus)

Stem-Bone Push-out Load Using ABAQUS

The Oculus design group chose to find the relationship between percent contact area (assuming at 0.5 mm interference) and push-out load using the finite element approach. ABAQUS finite element software was used to do this. In order to have variable contact area, the cross-section of the bone had to be noncircular, therefore; we chose the femoral bone intramedullary canal cross-section to be elliptical (Figure 4).

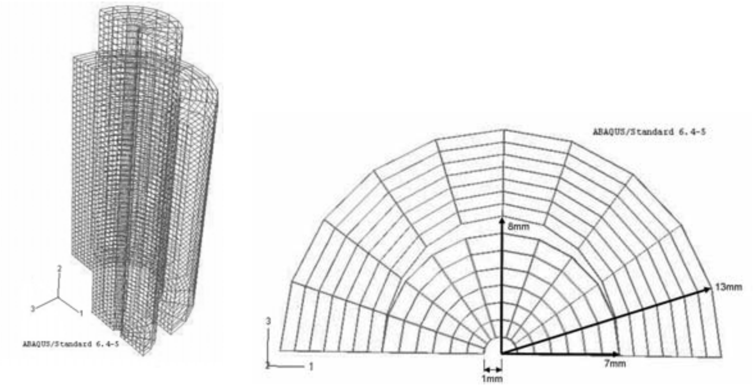


Figure 4: Elliptical bone model and Cross-section Layout. Only half the model is shown.

A 3D solid 8 node linear brick element with reduced integration and hourglass control, C3D8R, was used for both the bone and the stem. The stem was a cobalt-chromium alloy, the cortical bone was assumed to be transversely isotropic. The viscoelastic behavior was expressed by the time-hardening power law equation [4]. To have interaction, namely resolution of an interference fit and friction, surfaces needed to be created on the bone and the stem. Since the interacting parts of the bone and stem defined by elements are continuous and must be deformable, an element surface was used. The surfaces were created on the element faces inside the bone and outside the stem. To create the contact between the two surfaces defined, the surfaces had to be associated as a contact pair, and the surface interaction had to be defined. The cortical bone surface was considered the slave surface, and the stem surface was considered the master surface.

Analysis

The analysis was split into three non-overlapping individual steps which occurred in chronological order. These steps were resolution of the interference, application of the load with elastic response and application of the load viscoelastic response (Figure 5). The resolution of interference step was a static stress analysis where the contact interference and shrink parameters were used to bring the bone and stem to equilibrium with a change in geometry and reaction forces at the interface. The friction for the press-fit was based upon the Classic Coulomb Dry Friction Model. The states of the two surfaces were sticking and slipping, where slipping occurred when the shear stress had exceeded the product of the pressure between the two surfaces and the coefficient of friction. At a point where a material surface would yield by the shear stress, a maximum interface shear stress was defined to produce sliding independent of the contact pressure stress if the magnitude of the equivalent shear stress reached the maximum value. The final step of the analysis was the viscoelastic response to the applied load and the initial elastic behavior. The transient response was analyzed for a 24.0 hour period, which was the period of time for most of the creep and stress relaxation to occur based on the power law.

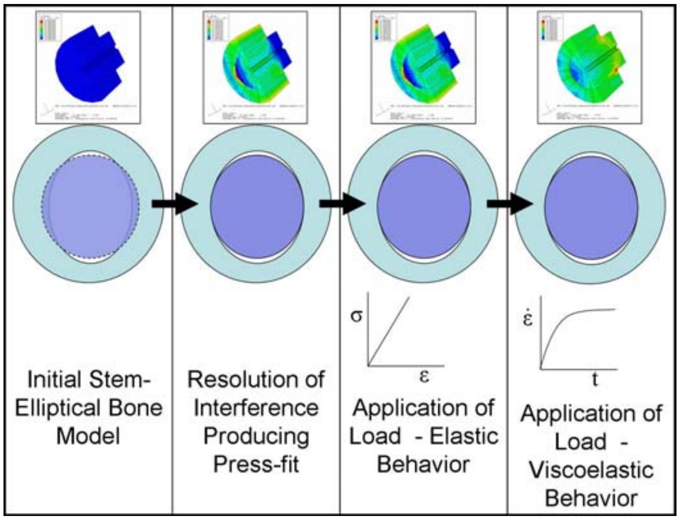


Figure 5: Diagram of SEBM model at various Analysis Steps.

RESULTS: Push-out Load Using ABAQUS

With the Stem-Elliptical Bone Model (SEBM), the input file was run for different nominal interference values which were constant if the canal was circular (using the APM model not discussed). This nominal interference was translated into contact area between the stem and canal wall. The push-out load was determined for various amounts of percent contact area and friction for elliptical bone (SEBM) and circular bone (APM). These data showed that the push-out load was significantly higher for contact area greater than 40% and interference area greater than 25%; however, there was not an advantage in push-out load if the canal was well lubricated with blood and bone marrow ($\mu \approx 0.15$). Accounting for viscoelasticity and less than 100%

contact area predicted in some cases thousands of Newtons less push-out load than ignoring these factors (Figure 6).

Correlations between contact area (generated by the Oculus) and push-out loads (calculated using ABAQUS) can be used to approximate the stability of an implant. These data also give surgeons immediate access to accurate contact information based on actual intramedullary canal measurements and push-out load predictions based on ABAQUS FEA. The data could be used to aid the surgeon in determining optimal stem size.

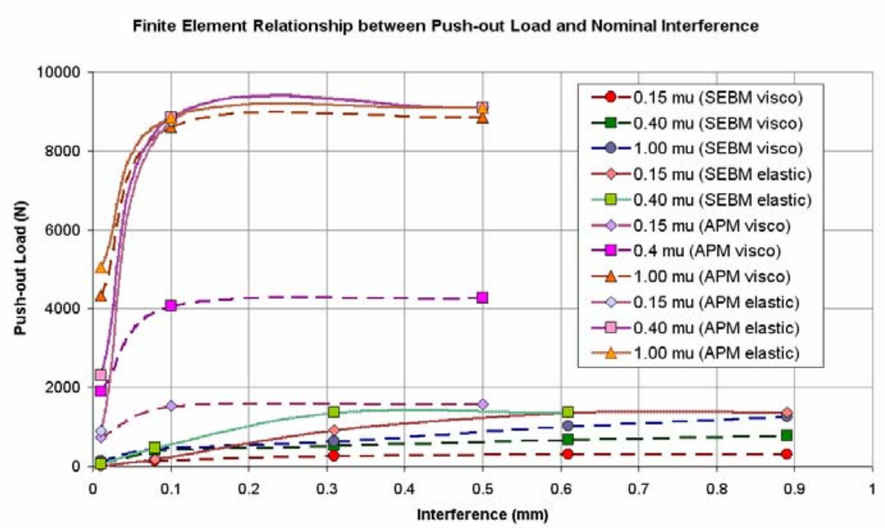


Figure 6: Relationships between push-out loads and stem bone interference. Comparisons are made between circular (APM) and elliptical (SEBM) model results and between viscoelastic and elastic material models. In the legend "mu" (μ) refers the coefficient of friction, "elastic" corresponds to elastic analysis and "visco" references to analysis that includes viscoelastic properties of bone.

Acknowledgement: This article was reviewed and edited by Genelle Schedlbauer, Cedarville University student.

References

[1] Norman, T.L., Ackerman, E.S., Smith, T.S., Gruen, T.A., Yates, A.J., Blaha, J.D., and Kish, V.L. "Cortical Bone Viscoelasticity and Fixation Strength of Press-Fit Intramedullary Rods: An In-Vitro Model," *J Biomechanical Engineering,* Vol. 28, pp. 13-17, 2006.

[2] Shultz, T.R., Blaha, J.D., Gruen, T.A., Norman, T.L. "Cortical Bone Viscoelasticity and Fixation Strength of Press-Fit Intramedullary Rods: A Finite Element Model"," *J Biomechanical Engineering,* Vol. 28, pp. 7-12, 2006.

[3] Blaha, J.D., "Press-fit components," *The Adult Hip* , ed. J.J. Callaghan, A.G. Rosenberg, and H.E. Rubash, Lippincott-Raven, Philadelphia, pp. 1085-1091.

[4] Brown, C.U., Norman, T.L., Kish, V.L., Gruen, T.A. and Blaha, J.D. "Time-Dependent Circumferential Deformation of Cortical Bone Upon Internal Radial Loading," *J. Biomechanical Engineering*, Vol. 124, pp. 456-461, 2002.