A THREE-DIMENSIONAL FINITE ELEMENT ANALYSIS MODEL OF AN ARTIFICIAL HIP AND BONE

J. A. Ecker
Applied Physics Laboratory
Aeronautics Department
The Johns Hopkings University
Laurel, Maryland

J. A. St. Ville
Department of Orthopedic Surgery
The Johns Hopkins Universtiy
Baltimore, Maryland

ABSTRACT

A three-dimensional finite element analysis model of an artificial hip implant in a cadaver femur has been developed to help further our knowledge of the stress distributions across the bone-to-implant interface in hopes of understanding the reasons for artificial hip failure (loosening) after only approximately seven to ten years of implantation. Actual measurements of the cortical and the cancellous bone were input to the model based upon computer axial tomography data. From these two-dimensional slices a three-dimension model was created representing the various layers of the bone and the implant. The cortical bone was modeled as a transversely isotropic material. Cancellous bone is known to be nonlinear and material properties are dependent upon location. This bone was modeled as totally anisotropic. Material properties for both bone types were taken from values previously reported in the literature. Different isotropic metals were modeled for the implant material. A polysulfone implant was also modeled. A standard 32mm acetabular ball was modeled at the end of the implant stem. The complete finite element model consisted of over 3100 nodal points and over 3000 isoparametric solid elements. Both hexahedron elements and pentahedron elements were used. Loading was applied over the surface of the acetabular ball to simulate pressures measured in vivo and reported previsouly in the literature. Abductor muscle loads were also included in the finite element analyses. This model enabled examination of the bone-to-implant interface in an analytical method. Variations in loadings, implant fixation, implant stem length and other considerations may be studied for sensitivity. We feel these types of studies may be instrumental in helping us understand the reasons for early failure of total hip arthroplasties.

INTRODUCTION

Due to the frequent failure of total joint arthroplasties due to fatigue or loosening, a large number of early revision surgeries are required. Computer representation of the physical interaction occurring in a total joint arthroplasty has been undertaken. A three-dimensional finite element model of a total hip arthroplasty has been developed to attain a better understanding of the influence of implant mechanical properties, material properties of bone and the bone-to-implant interface. This model allows for a great number of variations in modeling conditions to simulte changes in such parameters as material properties, geometries, loadings, fixation boundary conditions and implantation position. The usefulness of this model depends upon its ability to characterize real clinical issues, as well as, represent a true accurate model of the total human hip arthroplasty.

Clinical applications of finite element analyses have been quite broad. Prediction of implant failures, effects of various bone cements, length of implant stem and the relationship of normal forces to bone ingrowth have all been modeled with varying degrees of success. Over the past several years, both two- and three-dimensional finite element models have been developed. However, many of these previous studies had shortcomings because of inaccuracies in geometry, material properties, approximation of trabecular bone mechanics and representation of trabecular bone material constants. More recently, techniques have been developed to address these shortcomings and therefore, create a more representative model of the human hip.

This project seeks to minimize these inaccuracies by introducing such advanced concepts as gap elements to represent the changing bone-to-implant interface and more accurate modeling of bone regions with differing material properties. Even though use of these two concepts remains a compromise, a major advancement in the accuracy of the previous models should be realized.

Sensitivity studies will be conducted using the finite element model to determine the effects of varying implant stiffness, geometric shape, shaft length, implant fixation within the femur and loading conditions.

MATERIALS AND METHODS

A) Specimen Preparation

The femur of a human male weighing approximately 70kg was harvested from a cadaver and debrided of its soft tissue. The femoral neck was cut in a surgical fashion and a press-fit femoral component was introduced into the medullary cavity. The modified femur was then delivered to the radiology department and underwent computer axial tomography.

B) Computer Tomography

The femur was x-rayed using a Siemans Somatom DR3 and/or a GE 9800 CT-scanner. The scans extended from the isthmus of the femur to the proximal head and neck of the implant. The CT examination consisted of 35-45 overlapping transaxial scans obtained at 3mm intervals. The scanning parameters were machine dependent and were approximately 3 sec, 230mAs, 125UVp and 4mm collination. A two-dimensional image of each of the slices was created after an analog-to-digital conversion. The final signal processing was performed using a DEC PDP/11 computer system. Transparent hardcopy films were created and the image data were then transferred to 1/2-inch magnetic tape.

C) Pre-Processing of Data

The individual image data slices were read from the 1/2-inch magnetic tape using the PDA-PATRAN finite element pre- and post-processing software package from PDA Engineering. This software was used in the discretization of bone type and implant material for the two-dimensional image data. Nodal points were created at the borders of the different materials. Forty nodal points were created around the circumference of each of the borders. The two-dimensional image data arrays were then stacked to develop a three-dimensional arrangement of nodal points. Isoparametric solid elements were then created to provide connectivity between the nodes. A standard 32mm acetabular ball was modeled at the end of the implant stem to provide an appropriate loading base. The finite element model consisted of over 3100 nodal points and a total of over 3000 isoparametric solid elements. Both hexahedron (six face) and pentahedron (five face) elements were utilized to provide accurate shape adherence with the scan data. All of the pre-processing and model checkout computations were performed using PDA-PATRAN software on a SUN Microsystems 3/280C workstation and/or a Silicon Graphics 3000-series workstation.

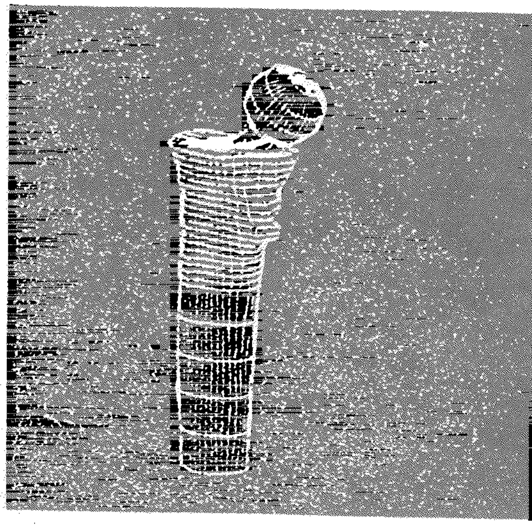

Fig. 1 Typical Finite Element Mesh

D) Finite Element Analysis

Finite element analyses were performed utilizing the general purpose MSC/NASTRAN software package from the MacNeal-Schwendler Corporation on the SUN 3/280C workstation. Input bulk data decks were directly created from the PDA-PATRAN models using the PATNAS translator from PDA Engineering with some manual modifications. Various advanced capabilities of the MSC/NASTRAN software were employed to simplify the analysis procedures. Internal nodal resequencing and optimization were performed to reduce execution time and to retain compatibility between different models. Checkpoint and restart capabilities were used to reduce formulation times of stiffness and mass matrices when only minor loading or constraint changes occurred. These capabilities were also used when outputing the nodal stiffnesses for the external calculation of gap element stiffnesses. Substructure (superelement) analysis techniques were used to partition the bone and the implant structures and the nonlinear gap elements to conserve computational resources and to increase solution efficiency.

E) Post-Processing

Results data were output from the MSC/NASTRAN finite element analyses for display and evaluation using capabilities of the PDA-PATRAN software. Alter routines and the OUTPUT2 command were used to output these data. Displacements, stress, strain and force data were then translated using the NASPAT translator from PDA Engineering. Element stress and strain results data were calculated at nodal points instead of at element centroids for use in evaluation because these data are between elements and are not averaged across the element volumes. This also resulted in saving time because nodal results distribution files need not be created when displaying results. Color-coded contour and fringe plots of the stress and strain results data on the finite element models were produced using the post-processing capabilities of the PDA-PATRAN software. Displacement contours and deformed geometry plots were also created using PDA-PATRAN. The post-processing was performed on the Silicon Graphics 4D workstation. All simulated animation was also performed on the SGI/4D workstation.

DISCUSSION

A) Modeling Techniques and Assumptions

Initially, a linear finite element analysis was performed assuming a perfect connection between the cancellous bone and the implant material. This analysis was conducted to represent a total hip arthroplasty with full bone ingrowth. Additionally, various combinations of only partially fixed implant models were analyzed. These analyses represented surgically implanted press-fit prostheses where contact with the cancellous bone was not complete. In the areas where space existed between the bone and the implant, nonlinear gap elements were created to model the changing bone-to-implant interface. Each gap stiffness was calculated from the MSC/NASTRAN nodal stiffness data for its two associated nodal points to guarantee model integrity and solution convergence. Further discussion of the clinical implications of the finite element models may be found in Part II of this paper.

B) Loads and Boundary Constraints

Pressure loads were applied to the acetabular ball to represent actual values measured in vivo (Hodge). The resultant of these pressures ranged from 100-2000 Newtons and the angular orientation varied depending on daily function (standing, mid-stance, stair-climbing, etc.) being modeled. Abductor muscle loads were applied to the surface of the greater trochanter. The resultant of these muscles have been reported in the literature to be approximately 300 lbs. with a moment of 2.1 inches, to the mechanical axis, at an angle 33 degrees (Hansen).

All rotations at the nodal points were constrained to a value of zero. Nodal points at the distal end of the femoral shaft were constrained in all six degrees of freedom. It was believed this set of full constraints were at a sufficient distance from the femoral head that stresses and strains in the critical region would not be significantly influenced.

C) Material Properties

The cortical bone structure was modeled as a transversely isotropic material and the mechanical properties for this paper have been taken from Carter (1978). The remaining cancellous bone structure was modeled as anisotropic. Different cancellous bone properties were used depending upon location within the model. These properties were based on work of Knaub, Evans, Vichnin and Cezayirlioglu. Several different metal materials were modeled for the prosthesis. These isotropically modeled metals included cobalt-chromium-molybdenum, stainless steel and titanium. A low modulus material was also modeled to represent a composite implant such as polysulfone. Again, this material was modeled with an isotropic material model. This was a simplification because no account was made for orientation of reinforcing fibers.

RESULTS

A generalized model of the hip has successfully been created. It allows for testing numerous practical clinical issues by varying boundary conditions.

This project introduced gap elements that are representative of actual conditions introduced by surgery. When an implant is placed within bone, there exists gaps (in some areas), between the implant and trabecular bone. At the gap a phenomena possibly described as 'micromotion' exists. This motion continues until the gap closes and then the two surfaces act as a continuum until the gap is re-opened. We feel this concept will be a useful tool to address the frequent unconstrained conditions that exist at the bone-implant interface.

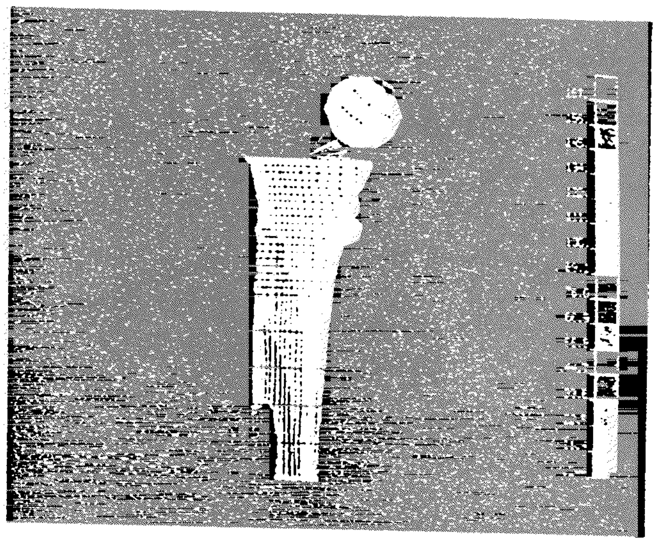

Fig. 2 Hencky-von Mises Stress Distribution at Cortical Bone Surface

A second issue addressed in this paper is the handling of the anisotropic properties of the cancellous bone. In most previous studies it was modeled as a continuous material property throughout. Since there is no way at present to model the complex and irregular geometry at cancellous bone, we have regionalized the material properties and have created superelements to address the regional variations. We recognize this is a compromise, but it is another step forward to improve the mathematical predictions of the complex properties of various areas within the model.

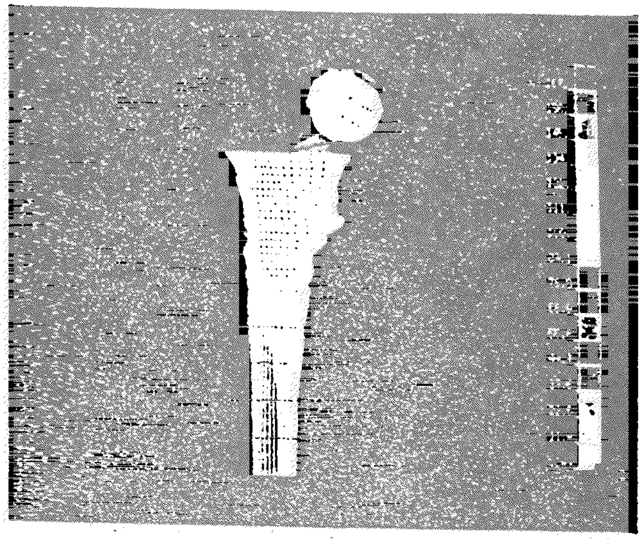

Fig. 3 Hencky-von Mises Stress Distribution at Cancellous Bone Surface

The final output of the model revealed very realistic stress contour data. Peak stresses were found on the lateral and medial cortices. The lateral cortex appears to support primarily tensile forces while the medial cortex possessed primarily compressive loads. As you move to the anterior and posterior cortex areas, it was found that the stresses approached zero (the transition point from tension to compression). This work corresponded precisely to the photoelastic research by L. Jones and D. Hungerford at Good Samaritan Hospital in Baltimore, Maryland.

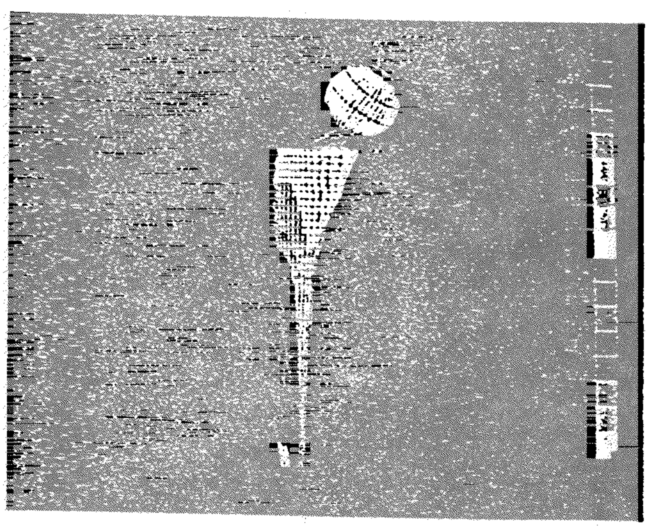

Fig. 4 Hencky-von Mises Stress Distribution at Surface of Co-Cr-Mb Prosthesis

REFERENCES:

I. Orthopedic Finite Element Reviews

A. Huiskes, R. and Chao, E.Y.S. (1983) "A survey of finite element analysis in Orthopedic Biomechanics: The First Decade." J. Biomechanics, Vol. 16, No. 6, pp.385-409.

B. Dietrich, D. (1987) " FE Analysis For the Body", Mechanical Engineering, March 1987, pp.48-52.

II. Composite Materials

A. Roberts, J.C. and Ling, F.F. (1980) "The Design of a Continuous Fiber Composite Surface Hip Replacement,"Transactions of the ASME, Journal of Mechanical Design, Feb. 1980, Paper No. 80-CA/DET-29, pp.1-7.

B. Ling, F.F., Roberts, J.C. and Jones, W.R. (1981) "Fabrication and Wear Test of a Continuous Fiber/Particulate Composite Total Surface Hip Replacement, NASA Technical Memorandum, #81746, Presented at Joint Lubrication Conference, October 1981, pp.1-14.

C. Koeneman, J.B., Hansen, J.M. and Weinstein, A.M., "The Effect of Interface Conditions and Implant Design on the Biomechanics of the Proximal Femur", ORS Transactions, 1988.

D. Lee, T.K., Magee, F.P., Weinstein, A.M., Emmanual, J., Johnson, R.H. and Hansen, T.N., "Retrieval of a Plasma Spray Porus Coated Femoral Hip Implant - A 3-Dimensional Quantitative analysis," ORS Transactions, 1987.

E. Koeneman, J.B. and Hansen, T.N., "Finite Element Analysis of a Composite Material Dog Femoral Component," ORS Transaction (Pending) 1990.

F. Roberts, J.C., Biermann, P.J. and Ling, L.L., "Preliminary Investigation of the Tribiological Characteristcs of Composite Coatings Sliding Against Ultrahigh Molecular Weight Polyethylene", Wear, 68(1981) pp.57-70.

III. Computer Tomography

A. Pfeifer, W.H., (1985) "Computed Tomography of Advanced Composite Materials," ASM/ESD Advanced Composites Conference December, 1985.

B. Menhardt, W. and Imme, M. (1989) "Contour Detection using Fuzzy Sets", Publication Pending.

C. Imme, M. (1989) "Semiautomatic and fully automatic contour detection in MR-images." Publication Pending.

IV. Femoral Stem Properties

A. Hansen, T.M. and Koeneman, J.B. (1987) "The Use of Structural Partitioning within ANSYS to Solve Non-Linearities within Human Hip Replacement Models", 1987 ANSYS Conference

B. Basu, P.K., Beall, A.G., Simmons, D.J. and Vannier, M.D. (1986) "3-D Femoral Stress Analysis Using CT Scans and p-Version FEM", Biomat., Med. Dev., Act. Org., 13 (3 & 4), pp.163-186 (1985-86).

C. Hansen, T.M. and Koeneman, J.B. (1988) "To Solve Nonlinearities with Human Hip Replacement Models", Finite Element News #4 (1988) August, pp.37-41.

D. Robinson, R.P., Lovell, T.P., Green, T.M. and Bailey, G.A., "Early Femoral Component Loosening In DF-80 Total Hip Replacements", (Publication Pending).

E. Crowninshield, R.D., Brand, R.A., Johnston, R.C. and Milroy, J.C. (1980) "An Analysis of Femoral Component Stem Design in Total Hip Arthroplasty," JBJS, 62-A, No. 1, January 1980, pp.68-78.

F. Koeneman, J.B. and Hansen, T.M. (1989) "The Effect of Elastic Modulus, Geometry, and Interface Conditions on Bone Stress Around Implants," (Publication Pending).

G. Huiskes, R. and Vroemen, W. (1985) "A Standardized Finite Element Model for Routine Comparative Evaluations of Femoral Hip Prostheses," ORS Transactions, 1985 p.98.

H. Torzilli, P.A., Uyle, R.F., Arnoczky, S.P. and Dueland, R.T. (1979) "Biomechanical Analysis of Two Canine Hip Prostheses," Journal of the American Animal Hospital Association,, Vol. 15, No. 2, March/April 1979, pp.185-190.

I. Rohlmann, A., Mossner, V., Bergmann, G. and Kolbel, R. (1983) "Finite-Element-Analysis and Experimental Investigation In A Femur with Hip Endoprosthesis," J.Biomechanics, Vol. 16, No. 9, pp.727-742, 1983.

J. Brown, T.D., Pedersen, D.R., Radin, E.L. and Rose, R.N. (1988) "Global Mechanical Consequences of Reduced Cement/Bone Coupling Rigidity In Proximal Femoral Arthroplasty: A Three-Dimensional Finite Element Analysis," J. Biomechanics Vol. 21, No. 2., pp.115-129, 1988.

K. Saejong, S., Hirano, S., Granholm, J.W. and Walker, P.S. (1987) "The Influence of the Interface on Bone Strains and Stem-Bone Micromotion in Press-Fit Total Hip Stems" ORS Trans., Jan. 1987.

L. Vichnin, H.H. and Batterman, S.C., (1986) "Stress Analysis and Failure Prediction in the Proximal Femur Before and After Total Hip Replacement," J. of Biomechanical Engineering, Feb. 1986, Vol. 108, pp.33-41.

V. Femoral Head and Neck

A. Rapperport, D.J., Carter, D.R. and Schurman, D.J. (1985) "Contact Finite Element Stress Analysis of the Hip Joint," J. Orthop. Research 3:, pp.435-446, 1985.

B. Bergmann, G., Graichen, F., Siraky, J., Jendrzynski, H. and Rohlman, A. (1988) "Multichannel Strain Gauge Telemetry For Orthopedic Implants," J. Biomechanics, Vol. 21, No. 2, pp.169-176, 1988.

C. Hodge, W.A., Figan, R.S., Carlson, K.L., Burgess, R.G., Harris, W.H. and Mann, R.W. (1986) "Contact pressures in the human hip joint measured in vivo," Proc. Nat'l. Acad. Sci, Vol. 83, pp.2879-2883, May 1986.

D. Rushfeldt, P.D., Mann, R.W. and Harris, W.H. (1981) "Improved Techniques for Measuring in vitro the Geometry and Pressure Distribution in the Human Acetabulum," J. Biomechanics Vol. 14, No. 5, pp.315-323.

E. Greenwald, A.S. and O'Connor, J.J. (1971) "The transmission of load through the human hip joint", J.Biomechanics, 1971, Vol. 4, pp.507-528.

F. Brown, T.D. and Shaw, D.T. (1983) "In Vitro Contact Stress Distributions in the Natural Human Hip", J. Biomechanics, Vol.16, No. 6, pp.373-384, 1983.

G. Brown, T.D. and DiGioia, A.M. (1984) "A contact-coupled finite element analysis of the natural adult hip," J. Biomechanics, Vol. 17, No. 6. pp.437-448, 1984.

H. Bergmann, G., Siraky, J. and Rohlman, A. (1984) "A Comparison of hip joint forces in sheep, dog and man," J.Biomechanics, Vol. 17, No. 12, pp.907-912, 1984.

VI. Material Properties of Bone

A. Reilly, D.T., Burstein, A.H. and Frankel, V.H. (1974) "The Elastic Modulus For Bone" J. Biomechanics, 1974, Vol. 7 pp.271-275.

B. Stone, J.L., Beaupre, G.S. and Hayes, W.C. (1983) "Multiaxial Strength Characteristics of Trabecular Bone", J. Biomechanics, Vol. 16, No. 9, pp.743-752, 1983.

C. Torzilli, P.A., Burstein, A.H., Takebe, K., Zika, J.C. and Heiple, K.G. "The material properties and structural properties of maturing bone," reprint from Mechanical Properties of Bone -AMD Vol. 45, Edited by S.C. Cowin.

D. Torzilli, P.A., Takebe, K., Burstein, A.H., and Heiple, K.G. (1981) "Structural properties of Immature Canine Bone," J. Biomechanical Engineering, November 1981, Vol. 103, pp.232-238.

E. Reilly, D.T. and Burstein, A.H. (1974) "The Mechanical Properties of Cortical Bone," JBJS, Vol. 56-A, No. 5, July, 1974, pp.1001-1022.

F. Cowin, S.C. (1986) "Wolff's Law of Trabecular Architecture at Remodeling Equilibrium," J. Biomech. Engineering, February, 1986, Vol. 108, pp.83-88.

G. Cowin, S.C. and VanBuskirk, W.C. (1986) "Thermodynamic Restrictions on the Elastic Constants of Bone", J.Biomechanics, Vol. 19, No. 1, pp.85-87, 1986.

H. Knaub, P. (1981) "Material Properties and Strength Behavior of Spongy Bone Tissue at the Coxal Human Femur," Biomed. Tech. 26 (1981) pp.200-210.

I. Cezayirlioglu, H., Bahniuk, E., Davy, D.T. and Heiple, K.G. (1985) "Anisotropic Yield Behavior of Bone under combined Axial Force and Torque," J. Biomechanics Vol.18, No. 1, pp.61-65, 1985.

J. Evans, F.G. (1973) Mechanical Properties of Bone, pp.83-121, Charles C. Thomas, Springfield, IL.

K. Fung, Y.C. (1981) Biomechanics, Mechanical Properties of Living Tissue, pp.383-400, Springer-Verlag, New York.

ACKNOWLEDGEMENTS

A) This project was funded partially by funds from the Johns Hopkins Applied Physics Laboratory internal research and development fund and the Institute for Bioengineering general research fund.

B) Specimen preparation and model generations were performed at Brighham and Women's Hospital, Orthopedic Biomechanics Laboratory, Boston, Mass. and at The Johns Hopkins

Hospital, Department of Orthopedic Surgery, Baltimore, Maryland.
C) Technical support and computer hardware contributions were from SUN Microsystems, Inc. and Silicon Graphics, Inc.
D) Computer Software charitable contributions were from The MacNeal-Schwendler Corp. and PDA Engineering.