Analysis of hip prosthesis designs using computational structural methods

Santosh Kottalgi[1], Gunjan Verma[1], Shitalkumar Joshi1 Marc Horner[2]

ABSTRACT

Total hip replacement (THR) is widely used nowadays to relieve patients from pain due to arthritis or fracture. A good hip prosthesis design leads to extended life of the joint and patient comfort. There are several mechanical (structural) aspects to the design of the prostheses. In vitro testing is costly and time consuming. In order to reduce the design time, finite element analysis (FEA) tools are used to simulate the prostheses and the joint. In FEA, the prosthesis is subjected to various loads and constraints to study the response under various steady positions and gait cycles. This paper showcases an FEA-based optimization process for designing a hip joint implant for maximum patient comfort quantified by the relative movement (sliding distance) between the implant surface and the bone tissues. A three-dimensional model of the adult hip joint is generated from a MRI scan image and simulated for finding the contact pressure and sliding distance between the implant surface and bone cavity. The entire procedure has been developed using the ANSYS Workbench platform.

INTRODUCTION

Hip replacement is a surgical procedure in which the hip joint is replaced by a prosthetic implant. Hip replacement surgery can be performed as a total replacement or a hemi (half) replacement. Such joint replacement orthopedic surgery generally is conducted to relieve arthritis pain or fix severe physical joint damage as part of hip fracture treatment. A total hip replacement (total hip arthroplasty) consists of replacing both the acetabulum and the femoral head while hemiarthroplasty generally only replaces the femoral head.

There are several known issues associated with the design of a prosthetic joint which can lead to significant pain and early failure if they are not addressed during the design phase. Some of the more common failure mechanisms include early loosening of the prosthesis, osteolysis and breakage of the Prosthesis. Each of these can be categorized as mechanical (structural) problems caused by inadequate structural design, which makes mechanical testing and analysis essential during the initial design phase of any joint.

Computational methods are now widely used for designing prosthetic implants. The advantage of these methods is that they are able to predict the response of an implant to the different forces applied by human activity. In particular, finite element analysis (FEA) provides flexibility to the designers while at the same time reducing the testing time as compared to the real prototypes. Computational modeling is also typically less expensive than in vitro testing. Computational modeling reduces the overall effort required for coming up with a new product, therefore reducing the cost of a prosthetic implant to the patients.

Yuichi et al[1] used FEA to examine the biomechanical characteristics of the femoral components of a resurfacing prosthesis. They used a three dimensional model to calculate the stress generated on the femoral head, a key contributor to neck fracture. Similar work has been done by several other authors. Leone et al[2] validated the results of their three-dimensional FEA model with experiments, emphasizing that computational models can be used to complement physical testing. Wear analysis of the joints has also been done using FEA codes. Saikko et al performed extensive work in simulating wear in the hip and knee implants.[3]

With the advent of the computational methods in designing hip prostheses a lot of work is also being done on the optimization of artificial joints. Optimization methods can also be used in conjunction with the biomechanical analyses to determine the best conditions for extended joint life. Optimization can provide solutions to improve the design and manufacture of the prosthesis.

The current paper summarizes the results of an optimization study of a hip prosthesis. A patient-based femur is used for the bone geometry and the femoral stem, material properties and loads are taken from the literature. The analyses are performed using ANSYS Workbench. ANSYS Workbench hosts the tools for the geometry creation, model development, computational solution and design optimization in a single platform.

[1] ANSYS India, 34/1, Pune Infotech Park, Hinjewadi, Pune, Maharashtra
[2] ANSYS Inc., 1007 Church Street Suite 250 Evanston, IL 60201, USA

Address of correspondence: Shitalkumar Joshi
ANSYS India, 34/1, Pune Infotech Park, Hinjewadi, Pune, Maharashtra Email:
Email: Shital.joshi@ansys.com

PROBLEM STATEMENT

A hip prosthesis replaces the joint between the pelvic bone and femur with an artificial one. The two components of the artificial joint are referred to as the acetabular cup and the femoral stem. This case study presents the use of a computational approach towards optimizations of generic femoral stem to obtain clinically admissible shape. According to Ruben[4], the sliding distance of the femoral stem in femur bone marrow is one of the major factors in the success of hip prosthesis. Stem sliding depends on the body weight distribution on the stem neck, width of the stem at the stem collar and friction between stem and bone marrow. Therefore, these factors are the focus of this study.

COMPUTATIONAL MODELING AND NUMERICAL METHODS

Entities exhibiting relative movement follow the basic law of motion. A study of the kinematics and dynamic responses of bodies under the action of various forces requires the solution of the equation of motion which is a partial differential equation by nature. Finite element methods are used to solve this equation. In this method, a virtual model is created which replicates the physical scenario. The model includes all the forces/loads acting on the geometry. It also includes the various physical constraints. Various loads acting on the joints can be due to the body weight, the motion of the patient, tension because the muscle pulls and other severe loads because of physical activities. A CAD software is used to generate the geometry. Several software are available in the market some of them are even capable to create a CAD geometry out of the X ray or MRI Scan images. These geometries can be used to generate a patient specific FEA models. Material properties for the bones as well as the implant are also provided as input to correctly predict the bio-mechanical response of the assembly. The output of a finite element analysis quantifies the deformations of the various components, the stress generated in the components, location of the areas susceptible to damage, wear patterns and other useful information. The numerical results are then correlated with the relative comfort level experienced by patient.

Following Ruben et al [4], a generic femoral stem geometry was created in the geometry tool ANSYS Design Modeler. Certain dimensions of the stem were parameterized for reference during the optimization procedure. The outer face of the stem was offset to create the bone marrow region. The stem geometry was combined with the Standardized Femur geometry v 2.3 [5] (this geometry is publicly available for download from the BEL Repository at biomedtown.org). This geometry includes separate cortical and trabecular zones and was cut at the collar for our work. Stem, bone marrow and femur are assembled for FEA as shown in figure 1(a) and 1(b).

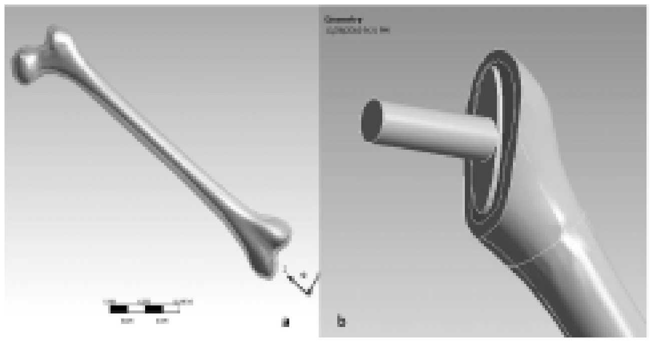

Figure 1: Stem and Femur Assembly

The computational mesh is generated by discretizing the geometry in Figure 1 into a set of smaller volumes called

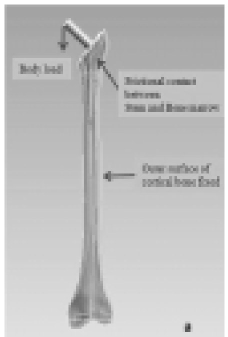

Figure 2a Mesh on the assembly

elements (see Figure 2a). The resulting mesh is made up of 95 thousand computational nodes .The equation of motion is then solved at the node points of each of these elements.

We now outline the boundary conditions and material properties used in the baseline simulation. Material properties of cortical and trabecular bone, bone marrow and stem [4] are provided in Table 1.

Table 1 Material Properties

Material	Young's modulus (GPA)	Poisson's ratio
Cortical Bone	17	0.3
Trabecular Bone	1	0.3
Bone marrow	1e-6	0.3
Stem	115	0.3

For boundary conditions, contact conditions between interfaces of various layers of bone and implant need to be specified along with external loads acting on the assembly. Contact between the stem and bone marrow is modeled as

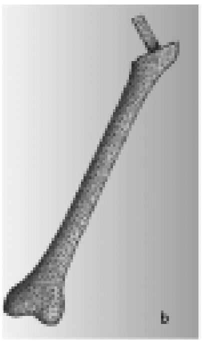

Figure 2b: Boundary conditions

frictional contact with a coefficient of friction of 0.3 in the baseline simulation. Rigid contact is applied at all other interfaces, which means there is no sliding between bone marrow and trabecular bone for example.

For the baseline case, body weight is considered to be 600 N which is equivalent to that of a young adult. The stem width at collar is 4.5 mm. With the given load and boundary conditions, finite element analysis is performed to analyze the design. The stress and stem sliding distance are evaluated as key factors contributing to patient discomfort. The acceptable limit of sliding distance for maximum human comfort is 30µm to 150 µm.[4]

SIMULATION RESULTS FOR THE BASELINE CASE

Steady state finite element analysis was performed using the baseline conditions outlined in the previous section. Simulation time for one loading conditions was about 1 hour on a Windows™ workstation.

Figure 3a shows the stress on the implant. As seen in the figure, higher stresses are seen towards the implant bottom. Stresses on the collar region of the implant were also noted to be high. Figure 3b shows the sliding distance between bone marrow and implant. Sliding distance is maximum in the downwards direction on sides of the implant. The maximum

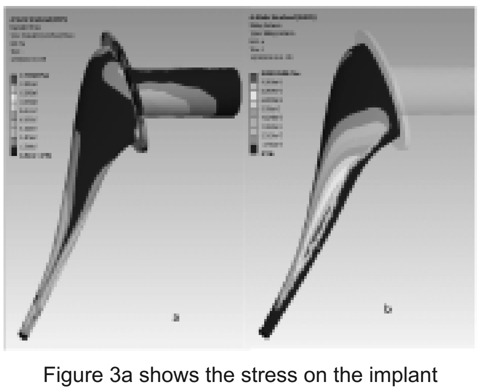

Figure 3a shows the stress on the implant
Figure 3b shows the sliding distance between bone marrow and implant

sliding distance is about 134 µm in this baseline case. This value is on higher side of the acceptable limit of 150 µm [4]. Sliding distance above150 µm will result in greater discomfort to the patient.

A design variation study indicated that the sliding distance increased with increased load. This is in line with the expectation that a heavier person is more likely to experience more discomfort.

Optimization

In this section, variation in the design parameters and their effect on sliding distance was studied using optimization techniques. Mathematically speaking an optimization problem has a goal or a target function which the user wants to achieve within the existing level of constraints. In the current study, the goal is to find designs that exhibit sliding distances within the specified limits of 30 to 150 µm.

The design space for the optimization problem was defined by the acceptable variation of the input parameters. The variations in input parameters are shown in Table 2.

Table 2 Input Parameter Variation

Parameter	Minimum Value	Maximum Value
Coefficient of friction between bone marrow and stem	0.1	0.6
Body Load	450N	850 N
Stem Width	4.55 mm	5.45 mm

A central composite design based method was used to generate a Design of Experiment (DOE) matrix. The DOE matrix is a set of points which completely represents the design space, and the response surface method is used to fit curves over the DOE results. Useful information can be deduced from response surfaces with a minimum number of FEA simulations. Figure 6(a) displays a three-dimensional response surface showing the variation of output parameter sliding distance with input parameters of coefficient of friction and stem width. It can be seen that the sliding distance increases as the stem width is increases. Similarly reducing the coefficient of friction also increases the sliding distance.

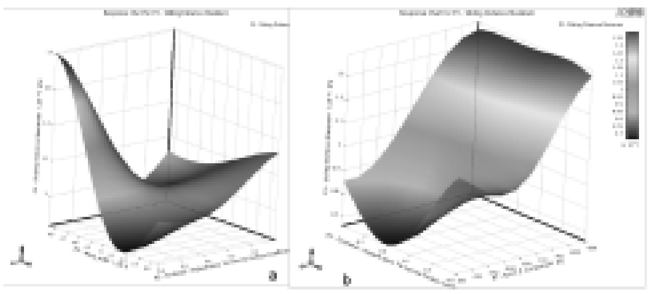

Figure 4(a) Response surface showing variation of sliding distance.
Figure 4 (b) Response surface showing variation of sliding distance.

The effect of stem width is more profound as compared to the coefficient of friction over the range studied. Similarly in figure 6(b) one can see that the sliding distance is more sensitive to the body weight acting on the joint.

The response surface gives a broad idea of how a design is going to behave under the given loading. In order to find combinations of the input parameters which give the desired goal of lower sliding distance and hence higher comfort, optimization analysis is performed.

ANSYS DesignXplorer provides the ability to perform goal-driven optimization. This method is based on the relative importance provided to various goals. Based on the response surfaces this method would find out the best designs from the multiple feasible designs. For the present analysis it was desired to find a design which would result in a sliding

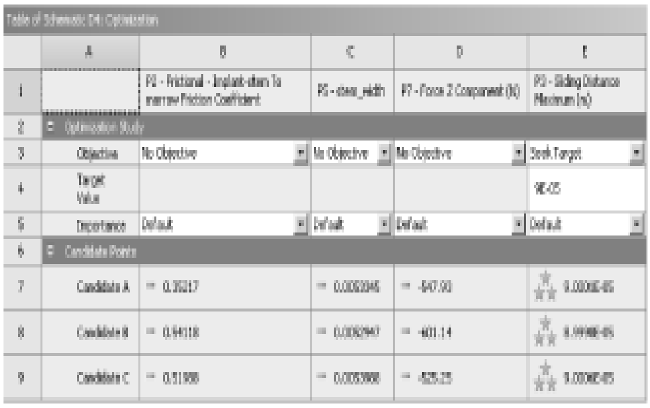

Table of Schematic D4: Optimization

	A	B	C	D	E
1		P? - Frictional - Implant-stem To marrow Friction Coefficient	P5 - stem_width	P7 - Force Z Component (N)	P3 - Sliding Distance Maximum (m)
2	Optimization Study				
3	Objective	No Objective	No Objective	No Objective	Seek Target
4	Target Value				9E-05
5	Importance	Default	Default	Default	Default
6	Candidate Points				
7	Candidate A	0.35217	0.0052045	-547.93	9.0000E-05
8	Candidate B	0.54118	0.0052947	-601.14	8.9998E-05
9	Candidate C	0.51988	0.0053888	-525.25	9.0008E-05

Figure 5: Inputs required for the optimization analysis

distance of 90 µm which is the mean value of the range considered to be acceptable [4].

Figure 5 shows the interface with the inputs required for the optimization analysis.

Candidate A was found to be more suited to our desired objective as it was closer to the specified goals. A direct simulation of the optimized design was carried out and was found to be in close agreement with the predicted outcome from optimization algorithm.

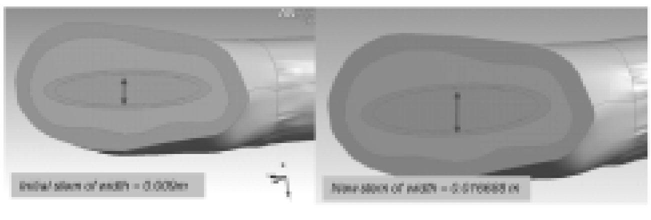

Figure 6 Optimization results.

Figure 6 shows the comparison between the baseline design and the optimized design.

CONCLUSIONS

Finite element analysis was used in this paper to understand and optimize the femoral component of a total hip prosthesis, with sliding distance being utilized as a key parameter that dictates patient comfort. A patient specific femur geometry obtained from a MRI scan was used in the current analysis. It was found that increasing body weight and stem width increased the sliding distance. Decreasing the friction coefficient between the stem and bone marrow also increased sliding distance. Response surfaces were then used to identify a stem width that would provide a specific sliding distance. A goal driven optimization was then performed to find best design among the feasible ones. Thus, it can be seen that using MRI scan images in conjunction with the FEA solver provides useful patient specific results. These results can also be used to fine tune the prostheses for maximum human comfort. This study can also be extended to analyze stresses to be evaluated as one of the criterion for determining patient comfort. In addition, non homogenous material properties of bone components can be used to increase the accuracy of simulation.

REFERENCES

1. Biomechanical study of the resurfacing hip arthroplasty, The Journal of Arthroplasty, Volume 15, Issue 4 , Pages 505-511, June 2000
2. Total hip arthroplasty-finite element analysis validation, Bioengineering Conference, 1990., Proceedings of the 1990 Sixteenth Annual Northeast
3. Slide track analysis of eight contemporary hip simulator designs, Olof Calonius, Vesa Saikko, Journal of Biomechanics 35 (2002) 1439–1450
4. Ruben R, Folgado J, Fernandes P, Shape optimization of uncemented hip prosthesis, Virtual and Physical Prototyping, Vol 1, No 3, September 2006, 147-15
5. Viceconti M, Casali M, Massari B, Cristofolini L, Bassini S, Toni A, The "Standardized femur program" proposal for a reference geometry to be used for the creation of finite element models of the femur. Journal of Biomechanics 1996;29(9):1241.

Source of Support: Nil, Conflict of Interest: none