Predicting implant stability: in vitro validation in artificial bone

Release 0.00

Thibaut Bardyn[1], Philippe Gédet[1], Wock Hallermann[2] and Philippe Büchler[1]

May 21, 2008

[1]MEM Research Center, University of Bern, Bern, Switzerland
[2]Department of Craniomaxillofacial Surgery, University Hospital of Bern, Bern, Switzerland

Abstract

Primary stability of osseointegrated implants is necessary for short and long-term success of the treatment. This paper presents a method to help clinicians preoperatively assess this primary implant stability. The method combines a planning software with a in-house finite element solver. Once the clinician has chosen a position for the implant on the planning tool, a finite element analysis is automatically started and calculates the mechanical stability of the implant at this position. The process is designed to be as simple and fast as possible for an efficient clinical use. Mechanical testing material was used to validate the stability measured by the software. The novel tool presented here leads the way to a new generation of intelligent computer-assisted tools able to give a priori indication on the life span of the implant.

Contents

1 Introduction

Osseointegrated implantology has been the most important innovation in dentistry for the last ten years. This technology is based on the biological bond that exists between the bone and the implant surface coating [1]. However, if the implant is not stable at the time of placement, micromotions may occur, which will result in a failure of the osseointegration [2]. This is even more problematic when immediate loading is chosen. Current methods for the assessment of primary implant stability such as Periotest, Resonance Frequency Analysis, insertion or removal torque are all postoperative [3]. Computer-assisted planning of implant placement is becoming popular among

practicians. It provides information on the position of critical structure such as nerves or blood vessels [4]. Nevertheless, mechanical information about the stability of the implant is missing. It would then be particularly useful to bring mechanical information when deciding where to place implants and thus help define their best possible position. Several attempts have already been made to generate patient specific finite element analysis before surgery. Fütterling [5] and Viceconti et al. [6] have designed a process that creates a patient specific finite element analysis after the planning for dental and hip implantology respectively. However, the process goes through the tedious creation of a specific mesh and involves the combined use of several softwares, making these methods ineffective for daily clinical use. Olsen et al. [7] suggested a quasi automatic method that combines in one package a planning tool with a patient specific calculation of the axial stability of the implant. Feasibility tests of this tool led to promising results. The work presented here improves this method by integrating a refined finite element solver in a computer-assisted surgery tool. The details of the concept as well as an in vitro validation in artificial bone are presented hereafter.

2 Methods

2.1 Description of the planner

The prediction of the primary stability of the implant is divided in four steps: planning of implant position, construction of the patient specific finite element model, finite element analysis and the clinical decision based on the analysis of results.

2.1.1 Planning of implant position

A planning software has first been implemented (figure 1). Using this tool the clinician can virtually manipulate the implant on the CT scan of the patient and thus avoid vital structures such as nerves or sinus. In the same time, direct rendering is used to give in real time 3D visualization of the patient's head and to determine the position of the implant in space. Information on the bone density extracted from the pixel values around the implant is given during the positioning.

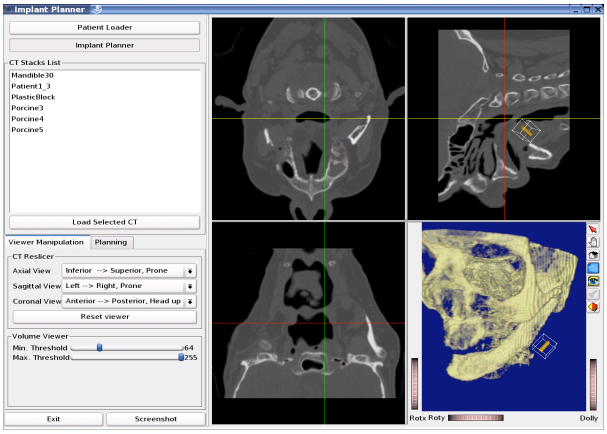

Figure 1 Screenshot of the implant planner. The position of the implant is shown both on the CT scan of the patient and on a volume rendering model.

2.1.2 Construction of the patient-specific finite element analysis

Once the position of the implant has been decided, the patient-specific finite element study may start. The mesh used for the simulation is a generic mesh representing the implant embedded in bone (figure 2). In this mesh, the implant is represented as a cylinder and threads are neglected. Particular attention is placed on the number of elements in this mesh in order to obtain the optimal speed to accuracy ratio.

For each patient, the mesh used varies only in terms of mechanical properties. These properties are extracted from the values of the voxels of the CT scan [8]. In this manner, patient specific information is transferred to the mesh. Elements outside of the bone are assigned near zero stiffness (0.0001 Pa), making the patient specific mesh construction entirely automatic. A Poisson's ratio of 0.33 was set for all elements.

This assumption was shown to have no influence on the resulting displacements [7].

Two different stimuli can be simulated in the planner: axial loading and removal torque. Each stimulus use different boundary conditions.

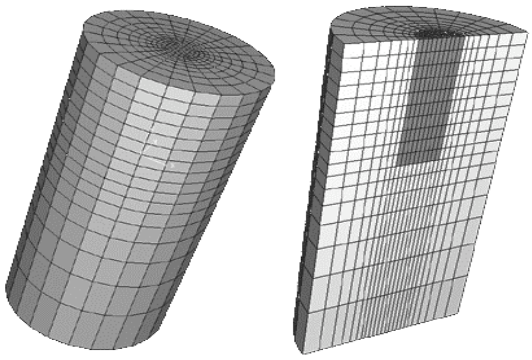

Figure 2 Generic mesh used for the finite element simulation. The mesh represents the implant (in dark grey) embedded in bone.

When an axial load is applied on an implant, the threads prevent any movement between the surface coating and the bone. Thus, a fixed bond between the bone and the implant can be assumed. A vertical force is applied on top of the implant and vertical displacement is calculated. The exterior of the mesh is fully constrained.

As for the removal torque, since the movement occurs in the direction of the threads, a frictional contact is considered between the implant and the bone. Contact properties depend on the radial pressfit forces that are exerted on the implant and the friction coefficient. However, generating and solving a finite element analysis with friction contact and pressfit would be too complex and too slow for a clinical use. Therefore a simplification of that problem is considered. In this simplified model, the contact between the two surfaces is not simulated. The removal torque is approximated by multiplying the friction coefficient by the total of the radial reaction forces due to pressfit at the bone/implant interface. This method limits the calculation to the simulation of the pressfit.

2.1.3 Solution of the finite element problem

For reasons of speed, the multifrontal method [9] was chosen for the solution of FE problem. The TAUCS [10] library was chosen for the inversion of the stiffness matrix and performs well in terms of accuracy and efficiency.

Once the patient specific finite element study is finished, the results are presented so that decision can be made whether to put the implant at the planned position or to try another location.

2.2 Validation using artificial bone

2.2.1 Preparation of the artificial bone blocks

Polyurethane foam blocks (Sawbones, Pacific Research Labotatories, Vashon, USA) were used as a substitute for bone in this study. This material has been tested with success in other implantology domains [11]. It is ASTM (American Society for Testing and Materials) approved and recognized as a “standard material for testing orthopaedic devices and instruments”. Its homogeneous properties make it an ideal material for comparative testing of bone screws. Foam blocks (54.9±0.3 mm length / 19.7±0.2 mm width / 14.9±0.3 mm height) were cut using a band saw. Blocks with two layers representing “trabecular” and “cortical” bone were used for this study. Two different densities were tested for each layer and two thicknesses for the top layer (figure 3).

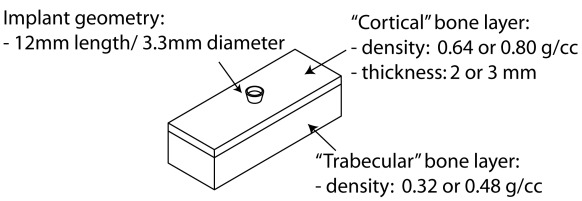

Figure 3 Different density and thickness combinations used in the experiments for the layered foam blocks and the geometry of the implants. Every possible combination was tested, resulting in 8 possibilities.

The implants tested were Straumann Standard© implants (Straumann AG, Basel, Switzerland) with a diameter of 3.3mm and a length of 12 mm. These implants were test implants and as such did not have any special coating. The insertion procedure recommended by the manufacturer was used to place the implants. The diameter of the pilot holes was 2.8 mm. A press drill was used to ensure the reproducibility of both holes direction and depth. The blocks were maintained at the same position and implants were inserted using a guide fixed to the drill to ensure a proper alignment with the axis of the hole.

2.2.2 Removal torque measurement

The blocks were embedded in dental plaster. The implant was attached via a gripper to a hydraulic testing machine (858 Mini Bionix, MTS, Minneapolis, USA). The gripper allowed free vertical displacement of the implant along its vertical axis during rotation. The embedded sample was placed in a metal container which had a load cell attached to its bottom. An alloy (Ostalloy 117, Metallum AG, Pratteln, Switzerland), with low melting temperature (47ºC), was poured in the container. After the alloy had solidified (≈10 min) it provided a rigid connection between the container and the embedded block. This protocol was used to ensure an exact alignment of the implant axis with the axis of the hydraulic testing machine (figure 4). A counterclockwise rotation of 30 degrees was then applied to the implant at a speed of 0.5 degrees per second and the reaction torque during the rotation was measured using a sampling rate of 20Hz. The "removal torque" was defined as the maximal torque (Nmm) measured on the torque/angle curve. Five blocks of each combination of densities were tested for statistical relevance.

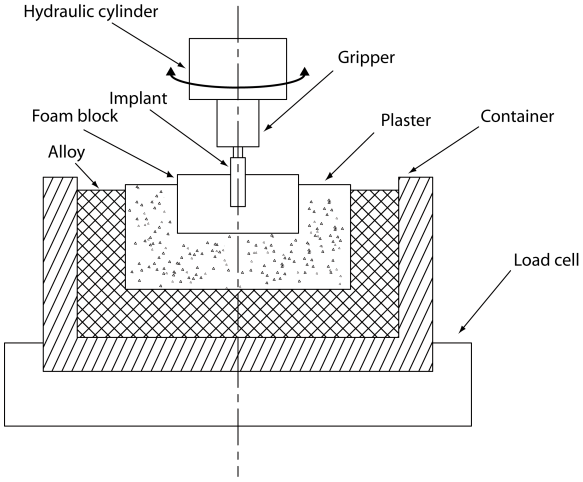

Figure 4 Protocol used for the removal torque test. The block with the implant was embedded in plaster and attached to the testing machine via a gripper. A counterclockwise rotation was then applied.

2.2.3 Axial loading measurement

The same setup as removal torque was used for axial testing except that after the solidification of the alloy the gripper was replaced by a flat loading plate. A ramp load up to a force of 200N was applied at a speed of 0.01mm per second to the implant. Maximal vertical displacement of the implant was measured during the test. Again, five blocks of each combination were tested for statistical relevance.

2.2.4 Comparison with finite element

Experimental results obtained for each test and each combination of density were compared with in-house finite element simulation. The Young's moduli of each material were known. In order to simulate removal torque test, friction coefficient between artificial bone and polished titanium was assumed to be 0.08 [12]. The inner diameter of the implants used in the experiments has the same diameter as the hole (2.8mm) while threads have a diameter of 3.3 mm. The pressfit distance was chosen so that the deformation of

the hole due to pressfit was equal to the volume of the threads.

For the axial loading simulation, a vertical load of 200N was applied on the top of the implant and resulting displacement was measured. A fixed bond was assumed between the implant and the bone.

3 Results

With the in-house finite element solver, the time required for the calculation of the displacements of the nodes is 12 s. A correlation of 0.9534 is found between experiments and simulation for the removal torque case. As for axial loading, a correlation of 0.8608 was found.

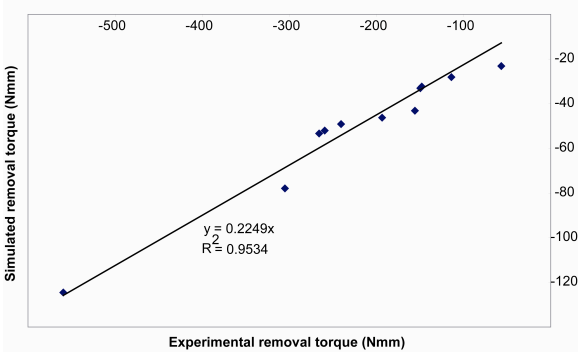

Figure 5 Correlation plot between the experimental and the numerical removal torque (in Nmm)

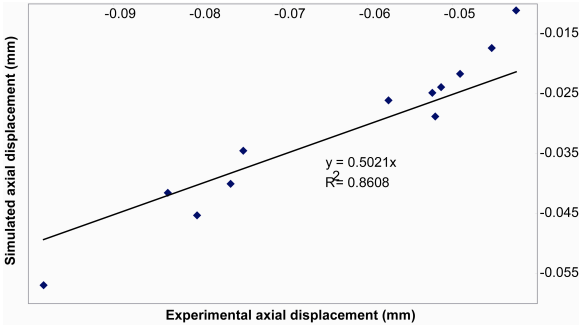

Figure 6 Correlation plot between the experimental and the numerical axial displacement (in mm)

4 Discussion

We present in this paper a method that gives mechanical information on the primary implant stability before the surgery has taken place. This is done by combining an in-house finite element study and a computer assisted surgery tool. The results obtained with the in-vitro validation show promising results in terms of accuracy and speed. The simple model used in this method will surely introduce some inaccuracy in the predicted output. Addition of transient remodeling law or threads on the implant model could improve the accuracy. However, both contact and remodeling laws are not well defined in the vicinity of an osseointegrated implant with the available clinical image data. In addition, such complex computation would compromise the interactivity of the tool and its use in a clinical environment. However, the final goal of this approach is not to exactly reproduce the mechanical situation of the implant but rather to provide a qualitative estimate of the total implant stability. Thus, comparison of the values among patients is possible and enables the distinction between normal and pathological cases. This concept offers a tremendous versatility as it can be applied to any existing planner as well as extended to other cementless implants (hip or knee for example). Further in vivo tests in sheep bone will help correlate the mechanical output of this tool to short and long-term results. This will result in a tool able to give to the clinician an accurate prognosis on implant life.

Reference

[1] P.I.Branemark. *Osseointegration and its experimental background*. J.Prosthet.Dent., 1983

[2] S.Szmukler-Moncler. *Timing of loading and effect of micromotion on bone-dental implant interface: review of experimental literature*. J.Biomed.Mater.Res., 1998

[3] N.Meredith. *Assessment of implant stability as a prognostic determinant*. Int.J.Prosthodont., 1998

[4] T.Fortin. *Computer-assisted dental implant surgery using computed tomography*. J.Image Guid.Surg., 1995

[5] S.Fütterling. *Automated Finite Element Modeling of a Human Mandible With Dental Implants*. The 6th Int'l Conf. in Central Europe on Comp. Graphics and Visualization '98 (WSCG '98)., 1998

[6] M.Viceconti. *Pre-clinical validation of a new partially cemented femoral prosthesis by synergetic use of numerical and experimental methods*. J.Biomech., 2001

[7] S.Olsen. *A novel computational method for real-time preoperative assessment of primary dental implant stability*. Clin.Oral Implants.Res., 2005

[8] D.R.Carter and Hayes, W. C. *The compressive behavior of bone as a two-phase porous structure*. J.Bone Joint Surg.Am., 1977

[9] J.W.H.Liu. *The multifrontal method for sparse matrix solution: theory and practice*. SIAM Review, 1992

[10] S.Toledo, Chen, D., and Rotkin, V. *TAUCS - A library of sparse linear solvers*. http://www.tau.ac.il/~stoledo/taucs

[11] S.Battula. *Experimental evaluation of the holding power/stiffness of the self-tapping bone screws in normal and osteoporotic bone material*. Clinical Biomechanics, 2006

[12] J.A.Grant. *Artificial composite bone as a model of human trabecular bone: The implant-bone interface*. J.Biomech., 2006