DENTAL MATERIALS XXX (2018) XXX–XXX

Available online at www.sciencedirect.com

ScienceDirect

journal homepage: www.intl.elsevierhealth.com/journals/dema

Fatigue lifetime prediction of a reduced-diameter dental implant system: Numerical and experimental study

Yuanyuan Duan[a], *Jorge A. Gonzalez*[b], *Pratim A. Kulkarni*[a], *William W. Nagy*[c], *Jason A. Griggs*[a,*]

[a] *Department of Biomedical Materials Science, University of Mississippi Medical Center, MS, USA*
[b] *Department of Oral and Maxillofacial Surgery, Texas A&M Health Science Center, TX, USA*
[c] *Department of Restorative Sciences, Texas A&M Health Science Center, TX, USA*

ARTICLE INFO

Article history:
Received 25 January 2018
Received in revised form
21 May 2018
Accepted 1 June 2018
Available online xxx

Keywords:
Cyclic fatigue
Accelerated lifetime testing
Finite element modeling
Dental implants
Fractography
Titanium
Simulated bone
Plastic strain
Brown-Miller criteria

ABSTRACT

Objective. To validate the fatigue lifetime of a reduced-diameter dental implant system predicted by three-dimensional finite element analysis (FEA) by testing physical implant specimens using an accelerated lifetime testing (ALT) strategy with the apparatus specified by ISO 14801.

Methods. A commercially-available reduced-diameter titanium dental implant system (Straumann Standard Plus NN) was digitized using a micro-CT scanner. Axial slices were processed using an interactive medical image processing software (Mimics) to create 3D models. FEA analysis was performed in ABAQUS, and fatigue lifetime was predicted using fe-safe® software. The same implant specimens (n = 15) were tested at a frequency of 2 Hz on load frames using apparatus specified by ISO 14801 and ALT. Multiple step-stress load profiles with various aggressiveness were used to improve testing efficiency. Fatigue lifetime statistics of physical specimens were estimated in a reliability analysis software (ALTA PRO). Fractured specimens were examined using SEM with fractographic technique to determine the failure mode.

Results. FEA predicted lifetime was within the 95% confidence interval of lifetime estimated by experimental results, which suggested that FEA prediction was accurate for this implant system. The highest probability of failure was located at the root of the implant body screw thread adjacent to the simulated bone level, which also agreed with the failure origin in physical specimens.

Significance. Fatigue lifetime predictions based on finite element modeling could yield similar results in lieu of physical testing, allowing the use of virtual testing in the early stages of future research projects on implant fatigue.

© 2018 The Academy of Dental Materials. Published by Elsevier Inc. All rights reserved.

* *Corresponding author at*: Department of Biomedical Materials Science, University of Mississippi Medical Center, 2500 N State Street, Jackson, MS 39216, USA.
E-mail address: Jgriggs@umc.edu (J.A. Griggs).
https://doi.org/10.1016/j.dental.2018.06.002
0109-5641/© 2018 The Academy of Dental Materials. Published by Elsevier Inc. All rights reserved.

1. Introduction

In recent decades, dental implants have been widely used as an effective method to replace missing teeth and improve the patient's quality of life. According to the facts and figures provided by the American Academy of Implant Dentistry, over 5.5 million dental implants had been placed by United States dentists as of 2006. Although most of the clinical studies reported overall five-year success rates of 93%–97% for dental implants, biological and technical complications are still frequently observed [1–5]. Technical complications of dental implants include abutment screw loosening or fracture, implant body fracture, and abutment and superstructure fracture [6]. In some cases, surgical intervention is required to remove the implant [7]. Mechanical fatigue is widely recognized as a major reason associated with the technical complications for metal dental implants because they are subjected to repeated masticatory load in the oral environment throughout the whole service life [8]. Fatigue failure manifests as three distinct stages: microscopic cracks initiated from the small imperfections in the area of stress concentration, subcritical crack growth, and final catastrophic failure after a crack reaches the critical stress intensity factor. Among these three stages, the crack initiation stage consumes up to 90% of the total fatigue lifetime of titanium and its alloys, yet there is usually no early sign to be noticed or detected during this stage until the final rapid fracture happens [9]. This can partially explain that technical complications are rarely observed in early implant failures but more frequently occur for the late failures. It is also worthy to note that the incidence of implant fracture is likely to be underestimated because long-term clinical studies for dental implants are limited, and a significant sponsorship bias has been detected in this area of the literature [10]. Since the significance of fatigue study of dental implants is widely recognized, a standardized fatigue testing protocol was developed in 2003 by the Organization for International Standardization (ISO 14801:2003) and revised in 2007. The ISO standard requires that the testing should be conducted at 2 Hz and carried out until failure or two million cycles in liquid media or conducted at 15 Hz until 5 million cycles in dry conditions [11]. By following this standard, testing for a single specimen could take approximately 11.5 days. Additionally, a relatively large sample size is needed for cyclic fatigue testing because lifetime data usually show much more variation than strength data. Apparently, fatigue testing of dental implants is very expensive in terms of both testing time and procurement of specimens, especially considering that the market develops so quickly, and there is less and less time for a new design to be finished and launched.

The finite element (FE) method is an efficient and well-established tool for mechanical study. It is a numerical method for stress and deformation analysis by discretizing a continuous structure into a finite number of elements. This method provides great capability to solve problems with complex geometry, which could not be solved properly by classical analytical methods, for example, anatomical structures of the human body [12–14]. For dental implants, FE analysis has been used to evaluate the biomechanical performance of various prostheses and loading conditions, investigate the stress distribution in the supporting bone tissue, and provide case-specific surgical planning and clinical outcome prediction [15–18]. In addition to the static stress distribution, the cyclic fatigue lifetime can also be possibly predicted with the help of a fatigue post-processor using the stress and strain results of FE analysis at a given load. This method has been frequently used in the automotive industry for fatigue design, durability analysis, prototype assessment, component testing and whole vehicle test control over the past 20 years. Modern fatigue models include stress-based life analyses such as von Mises analysis and maximum principal stress analysis, as well as local strain life algorithms such as maximum principal strain analysis, maximum shear strain, and Brown–Miller combined strain analysis [19]. Although FE-based fatigue analysis is commonly used for reliability studies in engineering, it has been relatively rare to be applied in the biomedical fields. Those FE-based fatigue studies which were validated by cyclic fatigue tests of physical biomedical specimens are even fewer. Oh et al. calculated the characteristic failure load of three-unit fixed partial dentures with different connector designs by using a finite element software (ANSYS, ANSYS, Inc.) in conjunction with a ceramic analysis and reliability estimation software (CARES/Life, NASA Lewis Research Center, Cleveland, OH) using the finite element results [20]. It was found that the experimentally measured rapid-fracture failure loads correlated well with those predicted by finite element simulation. Fischer et al. predicted the lifetime for dental ceramic bridges by using ANSYS for FE analysis and CARES/Life and suggested that it is a suitable tool to evaluate the lifetime expectations for different dental ceramic materials and bridge designs [21]. For dental implants, there have been several FEA-based fatigue analysis studies, but these results were not correlated with the cyclic fatigue data of physical specimens [22–24]. On the other hand, a few experimental studies were done to predict the lifetime of dental implant systems, but only failure location or strain were correlated with the results of the finite element models, not the lifetime data [25,26]. Therefore, in this study, we hypothesized that the lifetime of a reduced-diameter titanium implant system could be successfully predicted by using FE analysis with a fatigue post-processor. A comparison between the FE-based lifetime data and the cyclic fatigue data of this implant system under similar testing conditions was performed to test this hypothesis.

2. Materials and methods

2.1. *Finite element modeling*

A commercial 3.3-mm diameter titanium dental implant system (Standard Plus Implant NN, Straumann, Basel, Switzerland) was scanned using an X-ray micro-computed tomography (micro-CT) scanner (Skyscan1172, Micro Photonics Inc., Allentown, PA) with the pixel size of 4.25 μm. This implant system is made of CP grade 4 titanium with an external hex connection and a 3.5-mm implant/abutment platform. Fig. 1(a) shows the X-ray projection image of an implant specimen. Fig. 1(b) shows the transverse view of the CT scan after reconstruction. Transverse sections (n = 5882) were generated and processed using medical image modeling software (Mim-

Download English Version:

https://daneshyari.com/en/article/7857778

Download Persian Version:

https://daneshyari.com/article/7857778

Daneshyari.com