journal of prosthodontic research 62 (2018) 24–30

Contents lists available at ScienceDirect

Journal of Prosthodontic Research

journal homepage: www.elsevier.com/locate/jpor

Original article

In vitro fatigue tests and *in silico* finite element analysis of dental implants with different fixture/abutment joint types using computer-aided design models

Satoshi Yamaguchi[a,*], Yasufumi Yamanishi[b], Lucas S. Machado[c], Shuji Matsumoto[a], Nick Tovar[c], Paulo G. Coelho[c], Van P. Thompson[d], Satoshi Imazato[a]

[a] *Department of Biomaterials Science, Osaka University Graduate School of Dentistry, 1-8 Yamadaoka, Suita, Osaka 565-0871, Japan*
[b] *Department of Fixed Prosthodontics, Osaka University Graduate School of Dentistry, 1-8 Yamadaoka, Suita, Osaka 565-0871, Japan*
[c] *Department of Biomaterials and Biomimetics, New York University College of Dentistry,345 E 24th Street, New York, NY 10010, USA*
[d] *Department of Biomaterials and Biophotonics, Kings College London Dental Institute, Guy's Hospital, London, UK*

ARTICLE INFO

Article history:
Received 27 October 2016
Received in revised form 10 March 2017
Accepted 28 March 2017
Available online 17 April 2017

Keywords:
Bioengineering
Finite element analysis
Fatigue test
Dental implant-abutment design
Step-stress accelerated life tests

ABSTRACT

Purpose: The aim of this study was to evaluate fatigue resistance of dental fixtures with two different fixture-abutment connections by *in vitro* fatigue testing and *in silico* three-dimensional finite element analysis (3D FEA) using original computer-aided design (CAD) models.

Methods: Dental implant fixtures with external connection (EX) or internal connection (IN) abutments were fabricated from original CAD models using grade IV titanium and step-stress accelerated life testing was performed. Fatigue cycles and loads were assessed by Weibull analysis, and fatigue cracking was observed by micro-computed tomography and a stereomicroscope with high dynamic range software. Using the same CAD models, displacement vectors of implant components were also analyzed by 3D FEA. Angles of the fractured line occurring at fixture platforms *in vitro* and of displacement vectors corresponding to the fractured line *in silico* were compared by two-way ANOVA.

Results: Fatigue testing showed significantly greater reliability for IN than EX ($p < 0.001$). Fatigue crack initiation was primarily observed at implant fixture platforms. FEA demonstrated that crack lines of both implant systems *in vitro* were observed in the same direction as displacement vectors of the implant fixtures *in silico*.

Conclusions: *In silico* displacement vectors in the implant fixture are insightful for geometric development of dental implants to reduce complex interactions leading to fatigue failure.

1. Introduction

Marginal bone loss of dental implants occurs regardless of preventive measures [1] and despite the high success rates of dental implant therapy [2–5]. Marginal bone loss may be caused by surgical trauma [6], excessive occlusal loading [7], microbial contamination of the implant fixture-abutment microgap [8], fixture-abutment micromovement [9], and repeated screwing and unscrewing [10]. While a substantial amount of research has focused on these topics over the past three decades, a breakthrough engineering design that reduces these complex interactions leading to failure has yet to be elucidated.

To investigate the influence of mechanical stress/strain on the marginal bone of dental implants, the design of implant components has been explored. Internal joint types have been established as fixture-abutment joint types with greater fatigue resistance [11–13] and stability [14] and lower degrees of microgap [15] and bone loss [16,17] relative to conventional external joint types.

While positive results relative to external hexagon platforms have been reported, internal connection designs may present substantial design parameter variations that include the shape of the abutment, presence or absence of platform switching, and the shape of the implant fixture neck. Consequently, a clear reason for the mechanical stress/strain concentration of the marginal bone

* Corresponding author. Fax: +81 6 6879 2919.
E-mail address: yamagu@dent.osaka-u.ac.jp (S. Yamaguchi).

http://dx.doi.org/10.1016/j.jpor.2017.03.006

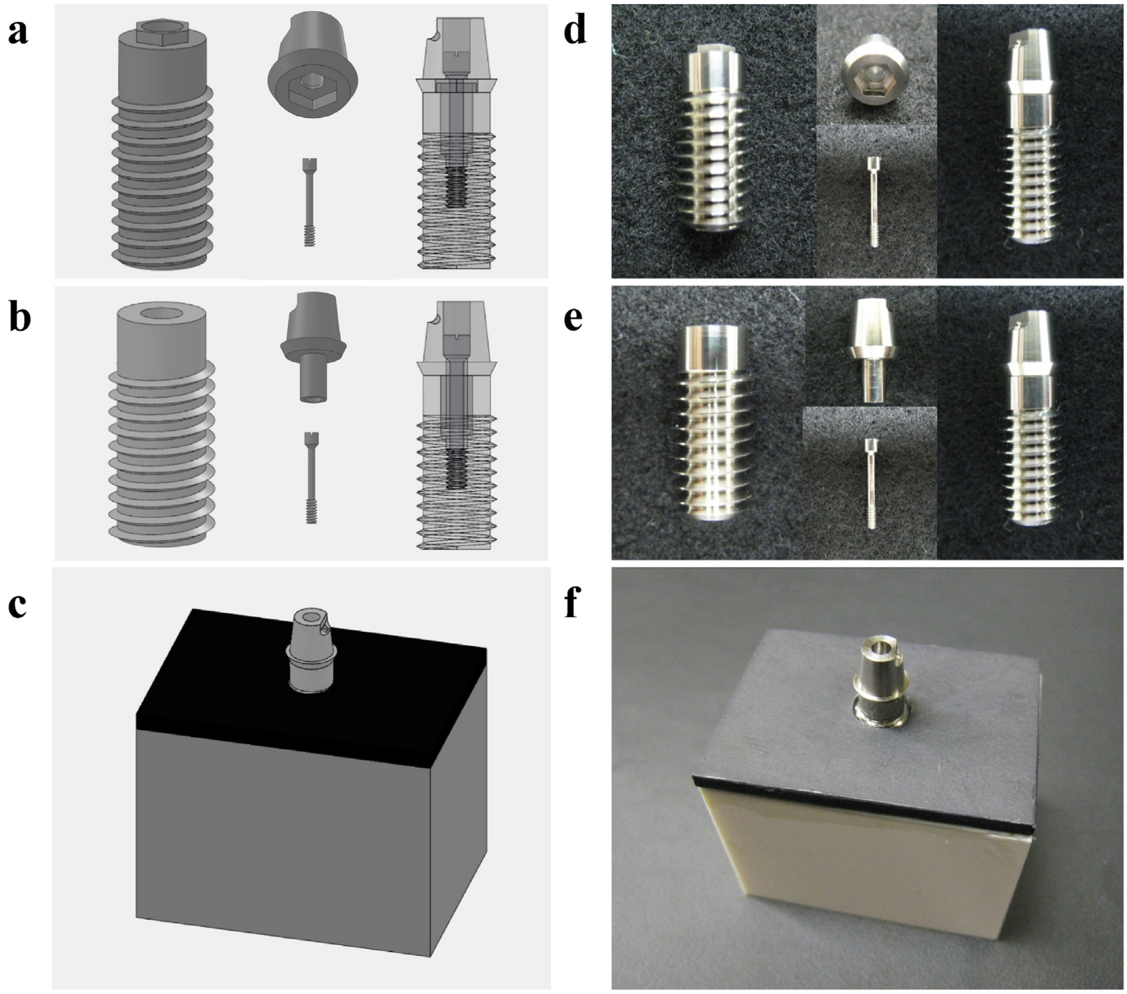

Fig. 1. Original CAD models and fabricated specimens. (a) CAD models of implant components (implant fixture, abutment, and abutment screw) and assembly with EX. (b) CAD models of implant components (implant fixture, abutment, and abutment screw) and assembly with IN. (c) Assembled model for 3D FEA. (d) Fabricated implant components from CAD models with EX and assembly. (e) Fabricated implant components from CAD models with IN and assembly. (f) Specimen for SSALTs.

has not been clarified, warranting optimization of multifactorial designs of implant components [18].

In vitro fatigue tests are one of the evaluation methods used to investigate the clinical reliability of dental implants. However, *in vitro* fatigue tests may fail to pinpoint the crack initiation site, even with the use of imaging techniques such as micro-computed tomography (micro-CT) or fractographic analysis. Therefore, step-stress accelerated life tests (SSALTs), which mimic mouth motion sliding contact, have been used to investigate the reliability of dental implants [19].

With this in perspective, *in silico* finite element analysis (FEA) is a useful approach to make comparisons focused on mono-factorial design [20] as it allows visualization and quantification of mechanical stress/strain distribution of the marginal bone of dental implant fixtures and implant components. Our previous work suggested that internal joint types are biomechanically more suitable than external joint types by using original computer-aided design (CAD) models [21]. Such results have been further supported by a number of *in silico* FEA studies using commercial dental implants containing mixed independent variables [12,22–24].

In this study, fatigue resistance of dental implants with two different implant/abutment joint types, the internal joint type and the external joint type, were evaluated by *in vitro* SSALT and *in silico* FEA employing our original CAD models. In addition, fatigue crack lines of the implant fixture *in vitro* and corresponding *in silico* displacement vectors were investigated.

2. Material and methods

2.1. Sample preparation

Dental implants with an external joint (EX) or an internal joint (IN) were machined from original CAD models (Fig. 1a–c) using ASTM grade IV titanium (GC Corp. Tokyo, Japan). The diameter and length (from the platform to the tip) of the implant fixture measured 5×13 mm, and the pitch of the threads measured 0.9 mm. The shape of the threads and abutment screw were the same in both implant types. The diameter and length of the abutment screw that connectedthe implant and abutment measured 1.5×11 mm (Fig. 1d and e). The implants were vertically embedded in resins with 3 mm clearance from the platform to the resin. Abutments were tightened to 20 N cm by a torque gauge (BTG50CN, Tohnichi Mfg. Co. Ltd., Tokyo, Japan). Resins mimicked mechanical properties of cortical and cancellous bones (polyphenylene sulfide: $35 \times 2 \times 27$ mm and polypropylene: $35 \times 25 \times 27$ mm, Ensinger Japan, Tokyo, Japan, respectively) (Fig. 1f).