Journal of Dental Sciences (2015) 10, 68–73

Available online at www.sciencedirect.com

ScienceDirect

journal homepage: www.e-jds.com

ORIGINAL ARTICLE

Mechanical properties of temporary anchorage device

Yi-Jen Liu [a†], Jenny I-Chun Sar [b†], Yen-Yin Chen [d], Yu-Ying Chiang [d], Eddie Hsiang-Hua Lai [a,c*], Weng-Pin Chen [d**], Chun-Pin Lin [a,c]

[a] Department of Dentistry, National Taiwan University Hospital, Taipei, Taiwan
[b] Department of Dentistry, National Taiwan University Hospital, Hsin-Chu Branch, Hsinchu City, Taiwan
[c] School of Dentistry, College of Medicine, National Taiwan University, Taipei, Taiwan
[d] Mechanical Engineering Department, National Taipei University of Technology, Taipei, Taiwan

Received 17 August 2013; Final revision received 2 September 2013
Available online 4 February 2014

KEYWORDS
finite element analysis;
mechanical properties;
temporary anchorage device

Abstract *Background/purpose:* In order to evaluate properly the effect of different geometric designs of mini-implants on their mechanical behavior, finite element analysis (FEA) has long been a popular tool. The aim of the present study was to set up a standardized mechanical experiment to validate the effectiveness and accuracy of FEA.
Materials and methods: Three commercially available mini-implants, Mondeal, Osstem, and Bio-Ray were inserted into artificial bone block with homogeneous density to remove the variability associated with bone. FEA and mechanical tests were performed.
Results: For the bending test, a 7.57% error was found between the mechanical test and FEA. For relative and maximum displacements, results from FEA were compatible with those from mechanical tests. The results of the relative displacement from FEA (Mondeal > Osstem > Bioray) were consistent with those from mechanical tests that Mondeal provided the greatest mean displacement before failure, followed by Osstem and BioRay. Furthermore, after simulating a 2-mm cortical bone layer in the FEA test, the pullout resistance increased for all three mini-implants, yet the variations in between decreased dramatically.
Conclusion: By incorporating FEA with real mechanical trial experiments, results from FEA have been validated and proved to be effective in studying the stress and strain distribution of mini-implants subjected to loading. FEA helps to evaluate how geometrical designs of mini-screws affect their clinical performance and may be useful in future improvement of

* Corresponding author. School of Dentistry, National Taiwan University, Number 1, ChangDe Street, Taipei 10048, Taiwan.
** Corresponding author. Department of Mechanical Engineering, National Taipei University of Technology, Number 1, Section 3, Chung-Hsiao East Road, Taipei 106, Taiwan.
E-mail addresses: eddielai0715@yahoo.com.tw (E.H.-H. Lai), wpchen@ntut.edu.tw (W.-P. Chen).
† These two authors contributed equally to this work.

http://dx.doi.org/10.1016/j.jds.2013.09.004

screw designs. Based on our results, we have found that in clinical situations, the cortical bone layer plays an important role in the stability of the mini-implants.

Introduction

Mini-implant anchorage has gained its popularity in recent decades for its numerous advantages over traditional mechanics, including its versatility, simple treatment mechanics, reduction of the total treatment time, and the more predictable outcome.[1] However, despite the advantages mentioned above, practitioners come across adverse effects as well, including biological damage, inflammation, pain, and discomfort. Moreover, failures such as implant loosening, displacement, and breakage occur.[2–6]

A wide selection of mini-implants is available at present. As a result of the increased need, more systems are being developed and expect to be introduced to the market. It is important to distinguish between the different designs of the numerous available devices, because they are not a single entity. According to recent reports,[2] mini-screws have been gradually removed because of their mobility before or during orthodontic force application. Thus, understanding which variables are related to this mobility and improvements of screw designs are necessary to help resolve the problem. Unfortunately, detailed research in this field is still in its infancy.

According to a previous study,[7] the average success rate of mini-implant is about 83.8%; far below that of dental implants. Yet, it has been identified that if we pay attention to refining the geometrical parameters of the implant design and achieving greater primary stability, success rate could be dramatically improved. When it comes to geometric issue, finite element analysis (FEA) is a useful tool to resolve structural problems.[8] By use of numerical methods, this technique divides the problem domain into a collection of many smaller and simpler domains (elements) in which the field variables can be interpolated with the use of shape functions.

Although FEA has brought many advantages, the disadvantages of computer solutions must be kept in mind when using this and similar methods. FEA does not necessarily reveal how the stresses are influenced by important variables such as material properties and geometrical features, and errors in input data can produce incorrect results that may be overlooked by the analyst.[9]

The aim of the present study was to set up standardized laboratory equipment and procedures to analyze the mechanical behavior of mini-implants. Furthermore, the effectiveness and accuracy of FEA were validated by means of real mechanical experiments.

Materials and methods

FEA

Generation of FEA models

Three commercially available mini-implants, Mondeal (Mondeal Medical Systems GmbH, Tuttlingen, Germany), Osstem (Osstem Implant Co., Seoul, Korea), and Bio-Ray (Bio-Ray Biotech Corp., Taipei, Taiwan) were illustrated using computer-aided design software "SolidWorks 2005" (SolidWorks Corp., Concord, MA, USA) imported into MSC.Patran 2005 (MSC Software Corporation, Santa Ana, CA, USA) to generate a triangulated shell element mesh, and then transferred to MSC.Marc/Mentat 2007r1 (MSC Software Corporation) for further FEA (Fig. 1 and Table 1). A cancellous bone block was modeled around the mini-implant with all threads imbedded in bone, except the first thread at the top. The overall dimensions of this block were 20 mm in height, 8 mm in mesiodistal length, and 8 mm in buccolingual width. Both the bone and the mini-implant elements were assumed to be homogeneous, isotropic, and linearly elastic. The Poisson ratio of titanium alloy and cancellous bone was 0.3, and the Young's moduli were 110,000 MPa and 1370 MPa, respectively. For simulation of the pullout condition, an additional 2-mm cortical layer was modeled on the top surface of the cancellous bone block (Young's modulus 13,700 MPa).

Simulation of the pullout and bending conditions

The axial-loading (pullout) condition was an axial displacement of 0.01 mm applied to the head of the mini-

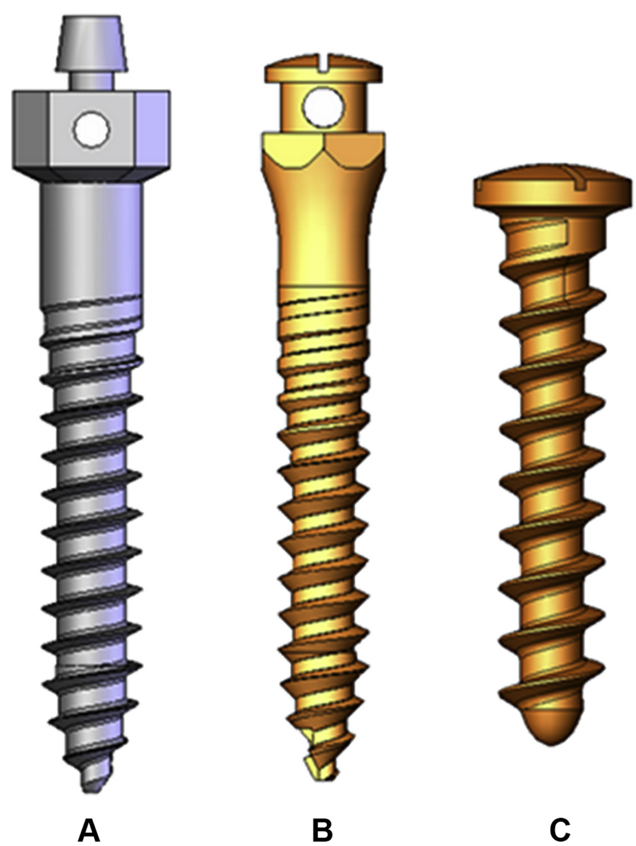

Figure 1 Models of three commercial mini-implants: (A) BioRay; (B) Osstem; and (C) Mondeal.