Biomedical Physics & Engineering
Express

PAPER

Mechanical evaluation of test configuration and dental implant geometry

To cite this article: L Pazos and E Perez 2018 *Biomed. Phys. Eng. Express* **4** 035024

Biomedical Physics & Engineering Express

RECEIVED
5 October 2017

REVISED
5 March 2018

ACCEPTED FOR PUBLICATION
8 March 2018

PUBLISHED
6 April 2018

PAPER

Mechanical evaluation of test configuration and dental implant geometry

L Pazos[1] **and E Perez**[2,3]

[1] Mechanics Research and Development Center, National Institute of Industrial Technology (INTI-Mecánica), Av. Gral. Paz 5445, (B1650KNA), San Martín, Buenos Aires, Argentina
[2] National Scientific and Technical Research Council (CONICET), Av. Rivadavia 1917, (C1033AAJ), Buenos Aires, Argentina
[3] Plastics Research and Development Center, National Institute of Industrial Technology (INTI-Plásticos), Av. Gral. Paz 5445, (B1650KNA), San Martín, Buenos Aires, Argentina

E-mail: eperez@inti.gob.ar

Keywords: dental implant, finite element analysis, mechanical behavior

Abstract

In the present work, the mechanical behavior of a dental implant system (screw-implant plus abutment) was studied. The effect of: (i) screw-implant geometry, (ii) contact condition and (iii) specimen holder material on the mechanical performance of a dental implant system was investigated. Finite Element Analysis and mechanical tests were carried out. The results showed that, screw-implant diameter and length affected the von Mises and maximum Principal Stress distributions. These geometrical factors influence the maximal values and stress concentration zone size, respectively. Maximal displacement values were drastically diminished with increased screw-implant diameter. Simulations indicated as probable failure locations the threaded surface of screw-implant and abutment, confirmed by mechanical tests.

1. Introduction

Dental implant systems used for single-tooth replacements generally consist of a screw-implant plus an abutment mechanically fixed by different connection designs, such as threaded abutment, retaining-screw, external hexagonal interface, internal connection, platform switching concept or Morse taper connection [1–6]. The dental implant mechanical performance has been related to the interfaces stability (implant-bone, implant-abutment), fatigue behavior, micro-movements, stress distributions into the implant system components and load transfer to the surrounding tissues, among a wide variety of factors [4–9]. Moreover, early clinical complications have been particularly reported for small implant dimensions or under severe loading conditions [4, 5, 8, 10–13].

With the aim of analyzing the implant behavior, a large variety of experimental, analytical and *in vitro* methodologies have been proposed [6, 8, 13, 14]. However, the relevance of many involved factors is still large under discussion. The standard ISO 14801 represents a general guidance for fatigue testing of dental implants. However, this standard does not provide experimental nor clinical information that indeed/effectively warrant the implant long-term success [15, 16]. Furthermore, the experimental set-up considered (specimen holder material, loading member geometry, loading condition, lubrication, implant tightening method among others parameters involved) can influence the mechanical behavior of tested implants [8, 17]. For this reason, a large quantity of tests should be carried out to determine the effect of each involved factor or to directly compare implant system designs. Due to this drawback, most of the published data is generally limited to a specific situation meaning that the comparison of results is not easy. In addition, some authors argue that, generally, manufacturers keep their experimental results strictly unpublished [6, 32]. For the analytical or simulated studies, the Finite Element Analysis (FEA) can be helpful to investigate the implant performance [18, 19]. Unfortunately, FE models involve simplifications or arbitrary definitions of some not really well known parameters such as: implant geometries and configurations, loadings and boundary conditions, contact surfaces, contact pressure distributions, elastic-plastic

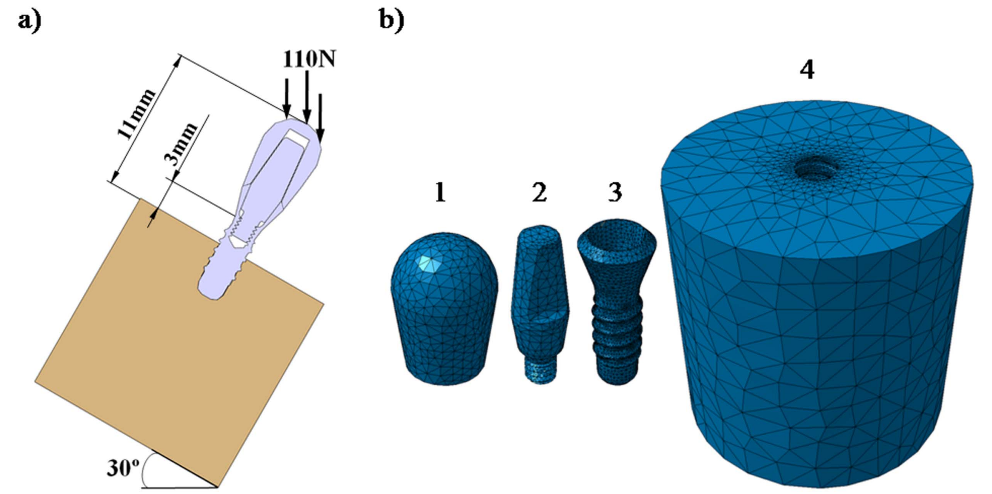

Figure 1. (a) Set up for mechanical tests and simulations, (b) FE model of the dental implant system (1: hemispherical loading member, 2: abutment, 3: screw-implant and 4: specimen holder).

material properties, among others. For these reasons, to avoid modeling mistakes and unrealistic simplification, experimental and simulated results should be compared to each other, and to clinical data, when possible [20–22].

Taking into account the previously published data, there is a brief description of some topics currently under discussion related to the mechanical performance of dental implants. Wu *et al* compared the mechanical performance of two dental implant systems (one-piece and two-piece) with small diameter (∅ 3.5 mm × 13 mm) [13]. The first one promotes a higher load transfer to the surrounding bone, suggesting a higher crestal bone retraction probability. The second is exposed to a higher stress level, meaning a higher risk of failure. In a similar way, Allum *et al* reported that the implant diameter (∅ 2.35–4.1 mm × 12–15 mm) drastically influences the implant yielding and fatigue performance [32]. However, in this work, generally, the dental implant system, geometry or material were simultaneously analyzed. Bicudo *et al* investigated two different implant-abutment connections (dimensions not reported) fixed in bone-like foams [9]. Displacement reduction was detected by both Morse taper system (higher dimension compared to external hexagon) and increased foam density. However, it would be interesting to analyze how the implant diameter and/or length variations influence this mechanical behavior. Minatel *et al* analyzed the implant geometry (∅ 4–5 mm × 10 mm) and connection (external hexagon, Morse taper) effects on the stress distribution [5]. For the external hexagon model, they observed reduced maximal stress into the screw-implant with larger diameter. On the other hand, the Morse taper induced lower stress and strain into the surrounding bone structure. Kayabasi *et al* simulated static and fatigue performance of a dental model (∅ 4.1 mm × 12 mm, threaded abutment) into a bone structure [18]. They concluded that the obtained low internal stresses should warrant the implant long-term success. Yang *et al* analyzed the implant-bone interface (bonded, no separation, frictionless) effect on the implant (∅ 3.7 mm × 15 mm, internal screw) and bone stress distribution [19]. They observed a peak stress reduction with bonded contact condition for both, the implant and the bone. In a similar way, Kong *et al* reported lower stress into the bone with increased implant diameter (∅ 3–5 mm × 6–16 mm) [26]. The whole simulated model was simplified as a single piece meaning that the implant-abutment connection effect has not been taken into account.

The aim of the present work is to analyze and directly compare the effect of: (i) implant geometry (thread profile, diameter and length), (ii) contact condition (screw-implant/specimen holder) and (iii) specimen holder material, on the mechanical behavior of dental implant system. This work was focused on an implant model extensively used, which we were allowed to publish by the manufacturer. In addition, a wide mechanical performance analysis was carried out by experimental tests and simulations of the dental implant system considered.

2. Materials and methods

A commercial dental implant system (FEDERA Implantes Dentales^{TM}) composed of a screw-implant and an abutment connected by an internal screw was analyzed. The set up used for mechanical tests and simulations are shown in figure 1.

2.1. Finite element modeling

The three-dimensional FE model was developed in ABAQUS 6.13-4. Table 1 shows the screw-implant dimensions (diameters and lengths) and the specimen holder materials (Pine wood, Titanium Grade 4) used.

Table 1. Dimensions and mechanical properties consideredfor the mechanical tests and the FE models.

	Screw-implant dimensions						
Component	Diameter (mm)	Length (mm)	Designation	Material	Elastic modulus (GPa)	Poisson rate	Yield Strength (MPa)
Implant	3.3	8	I-3.3-8 (non-threaded)	Titanium Grade 4	120	0.37	690 [9]
		8	I-3.3-8				
		14	I-3.3-14				
	4.8	8	I-4.8-8				
		12	I-4.8-12				
Specimen holder	20	20	….	Pine Wood	14 [23]	0.32 [24]	…
			….	Titanium Grade 4	120	0.37	690 [9]

The different implant systems were coded as: I-diameter-length. Dimensions of the other components were: ∅ 3.5 mm × 8 mm (abutment), ∅ 6 mm × 8 mm (hemispherical loading member) and ∅ 20 mm × 20 mm (specimen holder). Titanium Grade 4 was considered for all of the dental implant system components. All the materials involved were simulated as isotropic, homogeneous and linearly elastic (table 1). A load of 110 N with an inclination angle of $\theta = 30°$ (axial force: 95 N, transversal force: 55 N) was applied. The load value considered was similar to previously reported by other authors, while angulation was adopted from the standard guidance (ISO 14801) [15, 18–20, 25]. A distance of 3 mm from the nominal bone level was adopted, as suggested in the mentioned standard testing protocol to provide a representative situation of bone loss [15].

The FE model consisted of 135 500 10-nodes tetrahedral elements, including 95,500 elements and 143 000 nodes for the screw-implant. The mesh was refined on the threaded and contact surfaces. FEA with different mesh refinement levels were performed to obtain the results convergence. The model was considered to be convergent when the maximum of von Mises stress changed less than 3% [14, 26]. The contact condition was *bonded* between components of the dental implant system and specimen holder. In addition, for the screw-implant/specimen holder contact surface a second condition was analyzed (*friction*, coefficient of 0.3). For the boundary conditions, the whole model was fixed at the bottom surface of the specimen holder.

2.2. Mechanical tests

The I-3.3-14 dental implant system was tested (figure 1(a)) under quasi-static loading condition at a crosshead speed of 10 mm min^{-1} (Tinius Olsen H50-KT). Three samples were placed into specimen holders of Pine wood following the manufacturer recommendations. The implant-abutment and implant-holder connections were fixed with a torque of 45 Ncm and 32 Ncm, respectively. The distance of 3 mm, representative of bone loss, was checked by a micrometer. Fatigue tests were carried out over ten implants until failure or 2×10^6 cycles, whichever achieved first. A uniaxial sinusoidal cyclic loading ($R = 0.1$) with a frequency of 1.4 Hz was applied by a load-controlled machine (Instron 8874) following the standard recommendations of ISO 14801. All the tests were conducted in air at room temperature.

2.3. Tested samples characterization

The dental implant systems were observed before and after mechanical tests by scanning electron microscopy (SEM, FEI QUANTA 250). Fractured and threaded surfaces were observed at different magnifications to analyze the crack origin and propagation.

3. Results

3.1. Stress distribution analysis

The stress distributions for the dental implant systems are shown in figures 2–4 and listed in table 2. Figure 4 displays stress values collected along the screw-implant. The I-3.3-8 (non-threaded) implant exhibited a large von Mises stress concentration area (figures 2(a), 3(a), 4(a)) close to the upper plane of the specimen holder. In a similar way, for the screw-implant (I-3.3-8) stress concentrations were located on the firsts thread root (figures 2(b), 3(b)). For both contact conditions it was observed: (i) reduced maximal values with larger diameter and (ii) broader stress concentration zones with larger implant length. On the other hand, with *friction* contact condition (figure 4(d)) a shift of the maximal value to deeper threads was achieved. In general, similar variations were observed with both specimen holder materials (not shown here).

Taking into account the abutment stress distributions, the largest concentrations were observed on the firsts threaded root. The maximal values were reduced

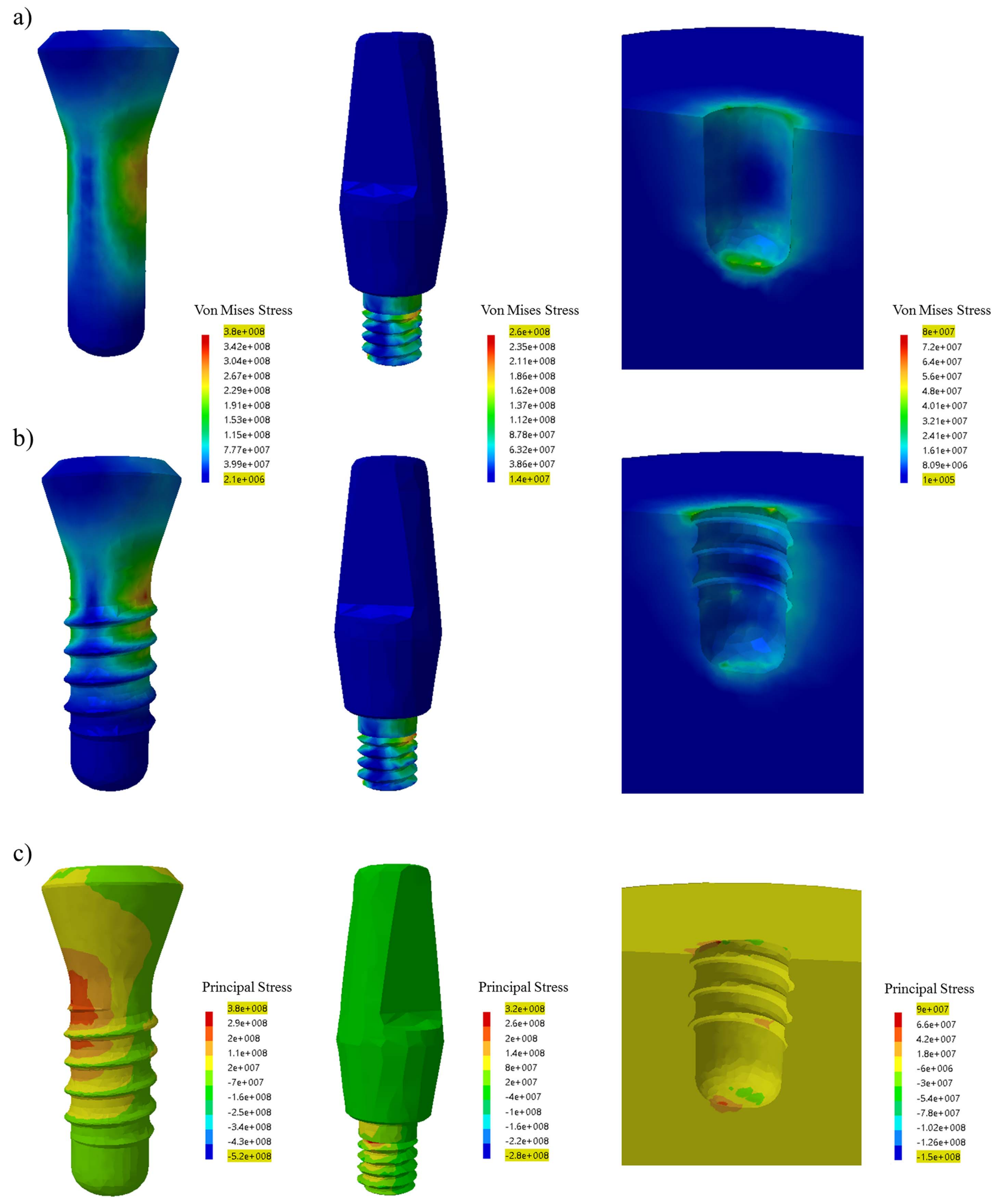

Figure 2. Stress distributions for *bonded* condition into Pine Wood: (a) von Mises stress of I-3.3-8 (non-threaded) system, (b) von Mises stress of I-3.3-8 system and (c) maximal Principal Stress of I-3.3-8 system.

with increased screw-implant diameter (abutment of: I-4.8-8 and I-4.8-12). Furthermore, for these systems supported by the Pine Wood specimen holder, a stress reduction into the abutment (*bonded*: from 81 to 44 MPa, *friction*: from 82 to 43 MPa) was exhibited with increased screw-implant length.

For the specimen holder, maximal values were observed on the threaded surface close to the upper plane. Furthermore, for the implant system with reduced diameter (specimen holder of: I-3.3-8, I-3.3-14) maximal values were increased with *friction* contact condition compared to *bonded* condition, regardless of the material. The largest values were achieved in the specimen holder of Titanium Grade 4 (*friction* contact) supporting the I-3.3-8 system.

The maximal Principal Stress distributions for the dental implant systems are shown in figures 2(c), 3(c) and listed in table 3. For the I-3.3-8 (non-threaded) system, the maximal tensile values were reached at the threaded surface of the abutment component. For larger diameter systems (I-4.8-8 and I-4.8-12) reduced maximal values were observed into the screw-implant and abutment. The most noticeable variations were detected into the abutment. For the specimen holder, the higher values were detected in the system with the smallest dimensions (I-3.3-8).

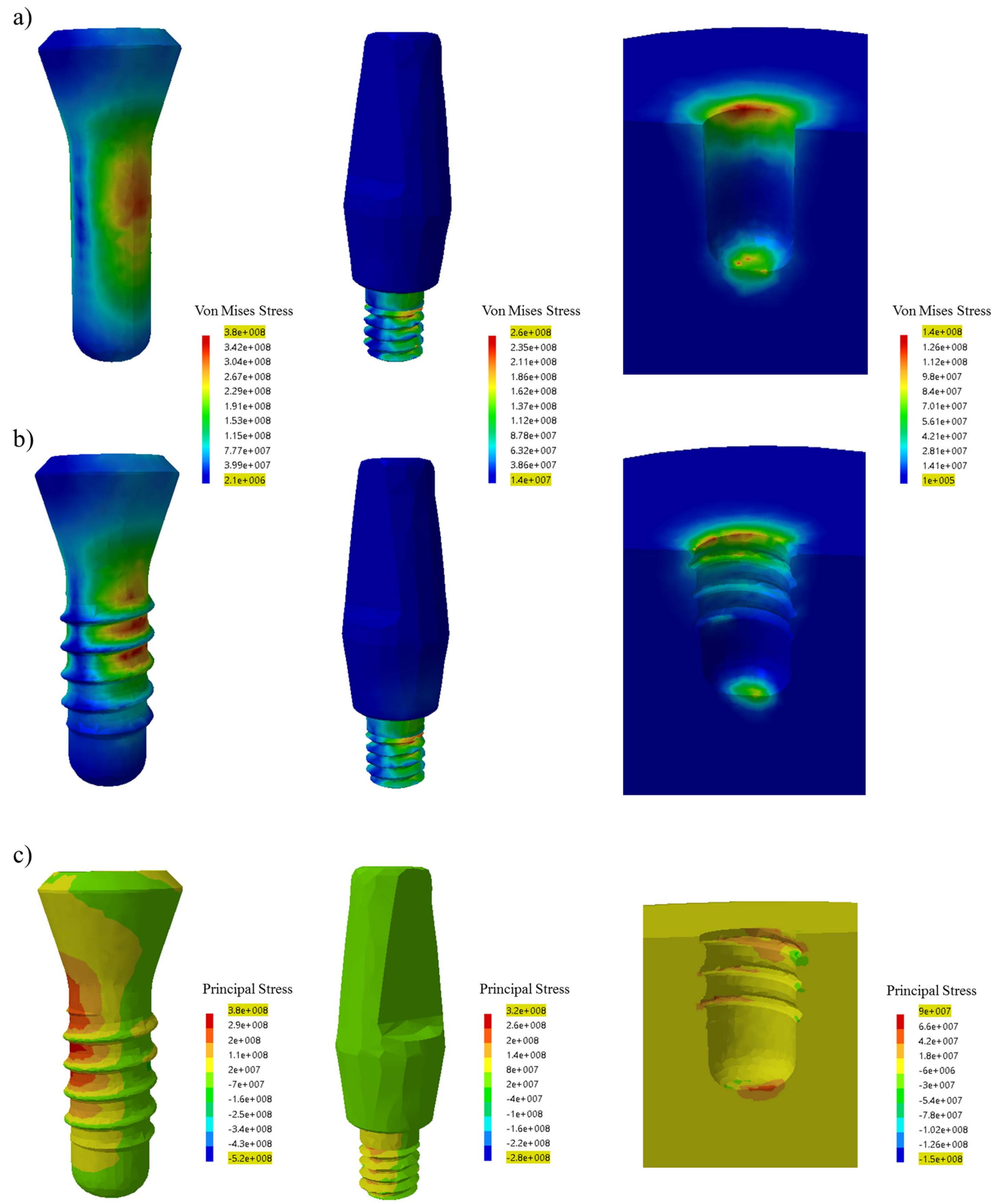

Figure 3. Stress distributions for *friction* condition into Pine Wood: (a) von Mises stress of I-3.3-8 (non-threaded) system, (b) von Mises stress of I-3.3-8 system and (c) maximal Principal Stress of I-3.3-8 system.

In general, the maximal Principal Stress distributions displayed irregular concentration zones close to the first thread root similarly to von Mises ones. On the other hand, these distributions also exhibited extreme values, related to tensile and compressive stresses not detected by von Mises stress, located on the opposite threaded surface sides. Lastly, FEA showed that maximal stress values did not reach the yield stress for any component of the dental implant systems and they were similar to published data [5, 13, 16, 17, 23].

3.2. Load-displacement relationship analysis

The load-displacement relationships and the maximal values, achieved with a load of 110N, are shown in figure 5 and table 2, respectively. The largest reductions of slopes and maximal values were observed with increased screw-implant diameter. The maximal displacement values were similar to previously reported by other authors [9, 19]. Furthermore, the effect of implant length (*bonded* condition) and specimen holder material (*friction* condition) were more noticeable with the smaller diameter systems.

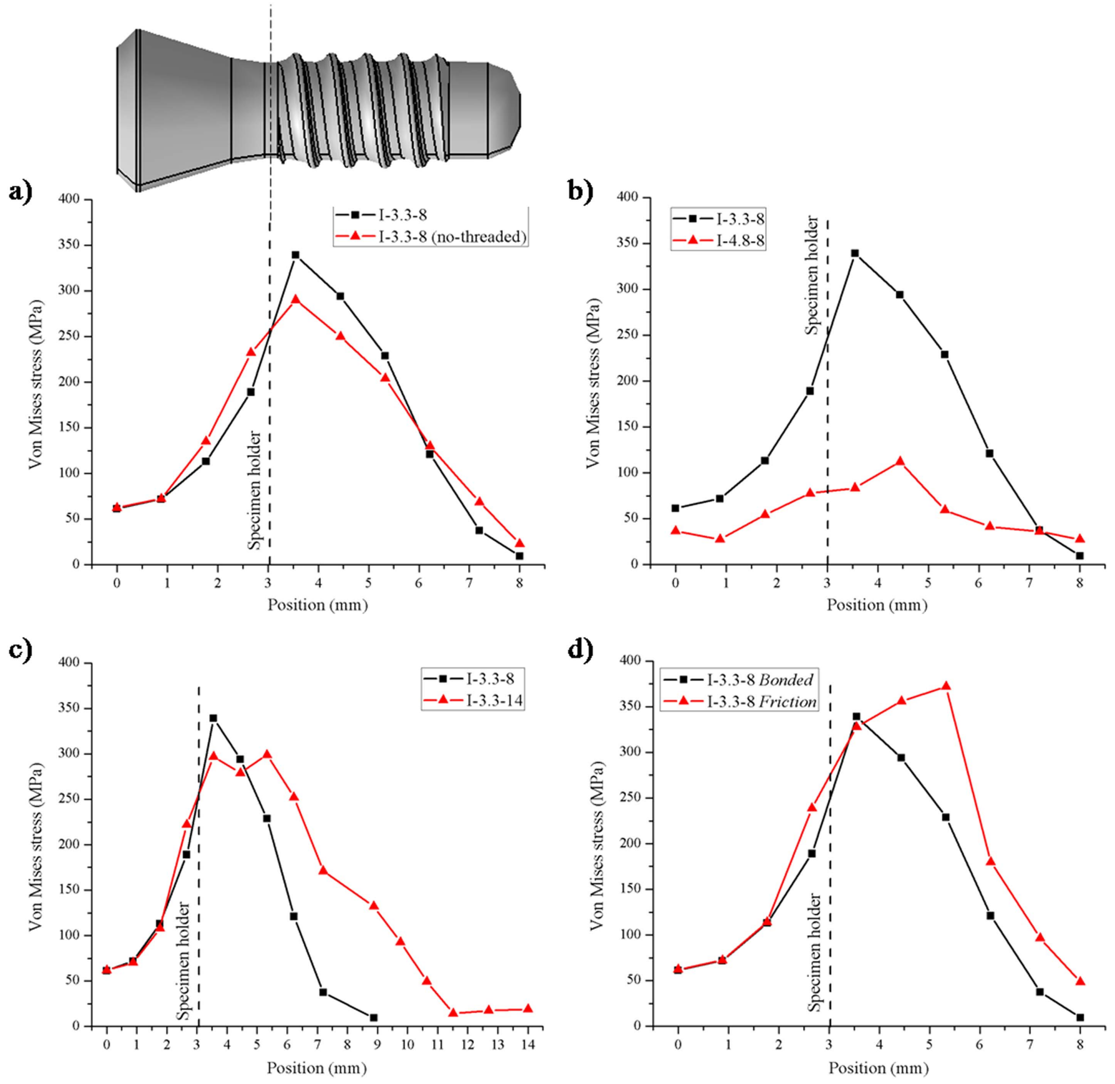

Figure 4. Comparison of von Mises Stress distributions along screw-implant for *bonded* condition into Pine Wood: (a) I-3.3-8 (non-threaded), (b) larger diameter (I-4.8-8), (c) larger length (I-3.3-14) and d) *friction* condition (I-3.3-8).

Table 2. Von Mises Stress and displacement maximal values for the different dental implant system.

		Maximal Von Mises stress (MPa)							
Implant	Holder material	Screw-implant		Abutment		Specimen holder		Maximal displacement (mm)	
	Contact condition	bonded	friction	bonded	friction	bonded	friction	bonded	friction
I-3.3-8 (non-threaded)	Pine	290	334	253	249	66	134	0.11	0.19
I-3.3-8	Wood	339	372	259	258	58	120	0.09	0.15
I-3.3-14		299	369	250	230	45	98	0.12	0.14
I-4.8-8		112	114	81	82	51	53	0.03	0.04
I-4.8-12		114	115	44	43	44	45	0.03	0.03
I-3.3-8	Titanium	341	354	257	253	118	215	0.08	0.12
I-3.3-14	Grade 4	298	368	250	244	79	196	0.11	0.11
I-4.8-8		119	122	42	43	105	104	0.02	0.02
I-4.8-12		118	119	40	41	103	96	0.02	0.02

3.3. Mechanical tests and fracture analysis

Figure 6 displays the experimental curves and tested samples of I-3.3-14 dental implant system. The quasi-static (figure 6(a)) tests displayed a linear elastic behavior until about 250N followed by no linearity. They were interrupted when a decrease of 5% of the maximum load was achieved. Experimental and simulated results (I-3.3-14, contact condition: *bonded*)

Table 3. Maximum Principal Stress values for the different dental implant systems.

Implant	Holder material	Maximum principal stress (MPa)					
		Screw-implant		Abutment		Specimen holder	
	Contact condition	bonded	friction	bonded	friction	bonded	Friction
I-3.3-8 (non-threaded)	Pine Wood	260	290	343	338	61	101
I-3.3-8		313	373	327	313	64	89
I-3.3-14		290	337	268	289	53	45
I-4.8-8		108	104	36	34	70	51
I-4.8-12		113	108	51	52	52	50
I-3.3-8	Titanium Grade 4	317	390	339	346	85	232
I-3.3-14		284	355	339	335	61	93
I-4.8-8		120	117	79	78	71	52
I-4.8-12		130	120	47	48	65	41

displayed similar linear behaviors suggesting the FEA accuracy. Figure 6(b) displayed the experimental fatigue curve obtained with different loading levels. As expected, it was observed that as loading values were reduced the number of cycles increased. For 200N, approximately, three samples reached the life criterion (indicated by arrows) suggested by the standard procedure. For an I-3.3-14 specimen fractured at 143 000 cycles (figure 6(c)), failure initiation was detected on the tensile side of the screw-implant, on the root located at the upper plane of specimen holder. All broken samples showed similar fracture patterns: crack initiation region followed by fatigue striations and failure zone. In an overall view, a smooth surface can be detected with a localized rougher region (failure zone). All the components (screw-implant and abutment) of dental implant systems were observed, before mechanical tests, by SEM and no defects were detected. On the other hand, SEM inspection (figure 6(d)) of fatigued dental implant systems did not reveal crack initiation on the abutment surface nor loosening of the dental implant system assembly.

4. Discussion

The effect of involved factors on the dental implant performance still remains under discussion. Particularly, geometry of the implant and interface stability are considered key factors for long term success close related to the biomechanical behavior. The relevance of implant diameter, length, thread profile, connection, contact area, material properties, among others have not been fully understood yet [5, 14, 20, 25–28]. In general, large diameters have been related to lower stress while the implant length effect has not been clearly established [9]. In addition, *in vitro* studies suggest the unsuitableness of small implant dimensions due to the large stresses and micromovements detrimental for stability and fatigue life [9, 13]. Experimental tests and FEA represent important tools to evaluate the close relationship between stress distribution and implants mechanical performance [18, 20].

In this study, FEA suggested that implant diameter affects the maximal stress values while implant length influences the stress concentration zone size. Specimen holder materials do not represent the complex bone structure nor its geometry, displaying a limitation of this study, but it allows a direct comparison of the stress transferred by the different dental implant systems to the surrounding structure. This kind of simplified analysis can be initially adopted to elucidate the implant design quality and the effect of involved factors. Furthermore, this limited analysis should be checked by experimental tests of dental implant into bone. For small diameter (I-3.3-8 and I-3.3-14) the length variation displayed a lower stress transferred to the specimen holder. It has been established that excessive stress can induce the crestal bone loss and early implant failure [18, 28]. In contrast, underloading of the surrounding bone can induce bone atrophy. This remarks the importance of keeping the transferred load between a certain range of values, but this range has not been clearly established in the literature yet [18]. The obtained results suggest a favorable mechanical performance of large screw-implant dimensions. The thread profile and contact condition can influence the load transfer too [29]. The studied models with smaller diameter (I-3.3-8 (non-threaded), I-3.3-8 and I-3.3-14), exhibited increased maximal values with the presence of the threaded surfaces while fully *bonded* contact condition displayed reduced values compared to *friction* condition.

In general, published information of implants' FEA is limited to von Mises stress analysis disregarding the Principal Stress distributions [12–15, 18–21, 28–30]. The importance of this analysis lays on the close relationship between tensile stress (it can be detected by the Principal Stress distributions) and crack initiation [8, 29–31]. The obtained results suggest probable failure locations on the tensile side of the screw-implant and abutment thread surfaces (not clearly detected by von Mises analysis).

The maximal displacement values with reduced diameter, displayed close similar variations for the screw-implant lengths or specimen holder materials

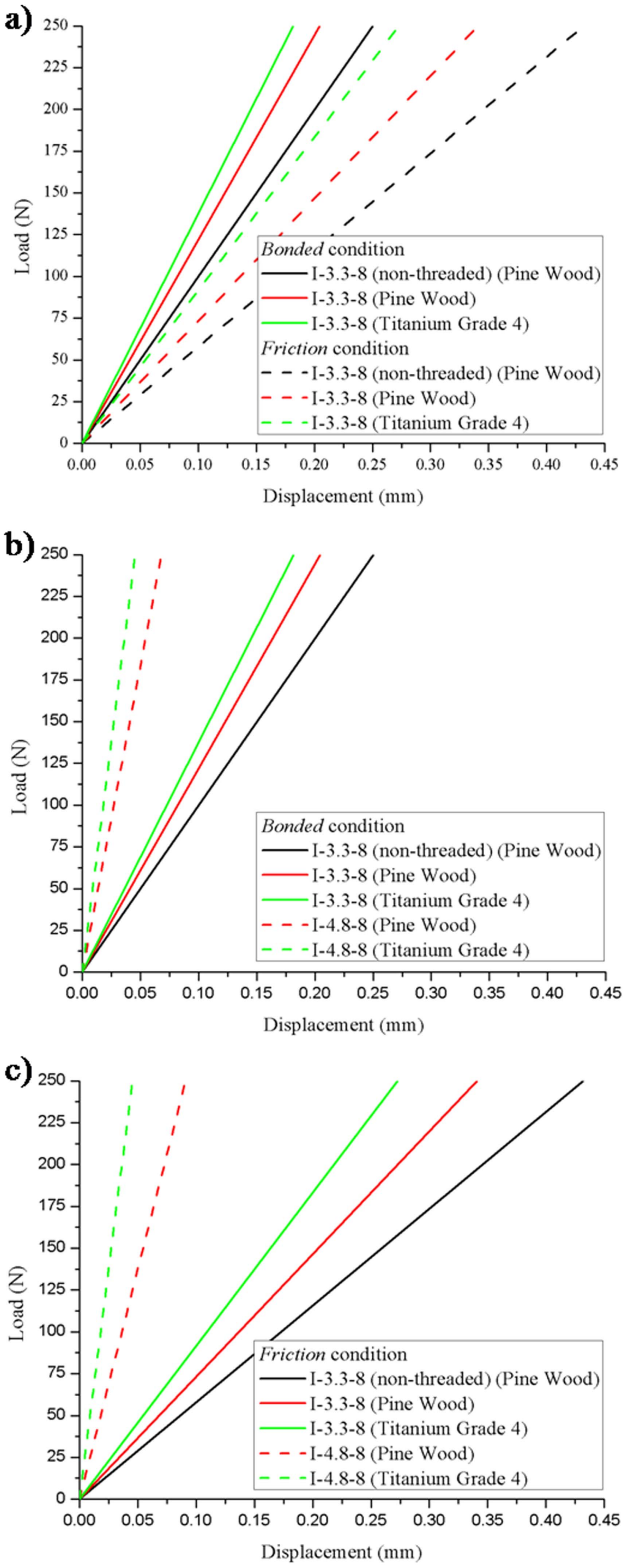

Figure 5. Comparison of load-displacement relationships for: (a) I-3.3-8 (non-threaded), larger diameter (I-4.8-8) (b) *bonded* condition and (c) *friction* condition.

studied. This observation suggests the relevance of the experimental set-up adopted. Comparison of reported result is generally not easy, as previously remarked. For example, when dental implant systems with different length and/or specimen holder material are compared, it should be checked if the displacement variation is strictly related to the screw-implant length or not. Otherwise, the rigidity variation due to the

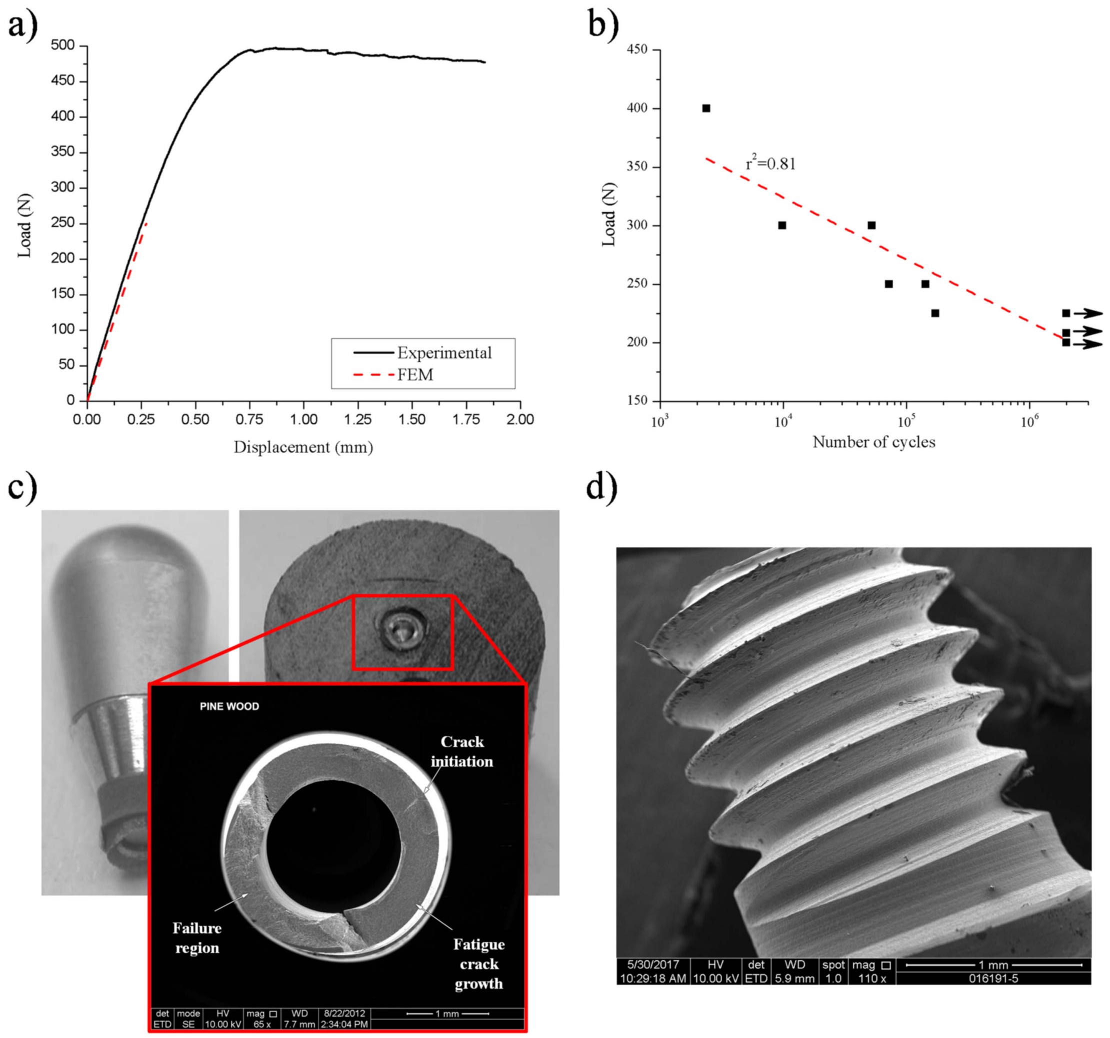

Figure 6. Mechanical evaluation of I-3.3-14 system into Pine Wood: (a) load-displacement curve of quasi-static test, (b) load-cycle curve of fatigue test, (c) screw-implant fatigued sample and d) abutment threaded surface.

specimen holder could induce misunderstanding of the experimental results. Furthermore, the maximal stress value into the specimen holder was obtained with Titanium grade 4 (I-3.3-8, *friction* condition). It suggests a higher probability to interrupt an experimental test due to the specimen holder failure.

Mechanical evaluations are simplified testing conditions assumed to be representative of the clinical situation. In addition, limited information can be found in the literature due to it been kept unpublished by manufacturers [6, 32]. Implants failure should be achieved by similar mechanisms, independently of the testing protocol and involved parameters, to warrant the *in vitro* validity. For this reason and taking into account that the specimen holder material influences the maximal displacement values, it could be considered an upper restriction on the elastic modulus in order to warrant the validity of the fatigue test results. For the fatigue tests performed, crack initiations were detected on the maximal tensile stress zone (screw-implant) and failure characteristics were similar to those previously reported [6–8].

5. Conclusions

The effects of implant geometry, screw-implant/specimen holder contact condition and specimen holder material were evaluated by numerical simulations and mechanical tests. The von Mises stress analysis displayed, with large implant dimensions, lower values into the dental implant system (screw-implant and abutment) and transferred to the surrounding tissues. Taking into account the effect of increased diameter or length, the first one promoted reduced maximal values while the second one a broader distribution. In addition, for reduced diameter a weaker screw-implant/specimen holder contact condition (*friction*) increased the maximal stress values. The maximum Principal Stress analysis indicates probable failure location on the screw-implant and abutment threaded surfaces.

The relevance of the experimental set-up has been well established in the literature. Particularly, the elastic modulus of specimen holder could be limited to warrant the experimental procedure validity (restricted influence on the rigidity variation and/or test interruption probability).

Acknowledgments

The authors of the present work want to acknowledge to FEDERA Implantes Dentales^{TM} for kindly providing the analyzed implant system, and to the National Scientific and Technical Research Council (CONICET) and to the National Institute of Industrial Technology (INTI) for financial support of this investigation.

ORCID iDs

E Perez https://orcid.org/0000-0001-9987-2850

References

[1] Quek H C, Tan K B and Nicholls J I 2008 Load fatigue performance of four implant-abutment interface designs: effect of torque level and implant system *Int. J. Oral. Maxil. Imp.* **23–2** 253–62

[2] Goodacre C J, Kan J Y K and Rungcharassaeng K 1999 Clinical complications of osseointegrated implants *J. Prost. Dent.* **81-5** 537–52

[3] Khraisat A, Stegaroiu R, Nomura S and Miyakawa O 2002 Fatigue resistance of two implant/abutment joint designs *J. Prost. Dent.* **88–6** 604–10

[4] Hoyer S A, Stanford C M, Buranadham S, Fridrich T, Wagner J and Gratton D 2001 Dynamic fatigue properties of the dental implant—abutment interface: joint opening in wide-diameter versus standard-diameter hex-type implants *J. Prost. Dent.* **85–6** 599–607

[5] Minatel L, Ramos Verri F, Abu Halawa Kudo G, de Faria Almeida D A, de Souza Batista V E, Araujo Lemos C A, Piza Pellizer E and Ferreira Santiago J 2017 Effect of different types of prosthetic platforms on stress-distribution in dental implant-supported prostheses *Mat. Sci. and Eng.* C **71** 35–42

[6] Wang K, Geng J, Jones D and Xu W 2016 Comparison of the fracture resistance of dental implants with different abutment taper angles *Mat. Sc. and Eng.* C **63** 164–71

[7] Karl M and Kelly J R 2009 Influence of loading frequency on implant failure under cyclic fatigue conditions *Dent. Mater.* **25–11** 1426–32

[8] Lee C K, Karl M and Kelly J R 2009 Evaluation of test protocol variables for dental implant fatigue research *Dent. Mater.* **25** 1419–25

[9] Bicudo P, Reis J, Deus A M, Reis L and Vaz M F 2016 Performance evaluation of dental implants: an experimental and numerical simulation study *Theo. and App. Fract. Mech.* **85** 74–83

[10] Gratton D G, Aquilino S A and Stanford C M 2001 Micromotion and dynamic fatigue properties of the dental implant—butment interface *J. Prost. Dent.* **85–1** 47–52

[11] Jemt T, Laney W R, Harris D, Henry P J, Krogh P H J Jr, Polizzi G, Zarb G A and Herrmann I 1991 Osseointegrated implants for single tooth replacement: a 1-year report from a multicenter prospective study *Int. J. Oral. Maxil. Imp.* **6–1** 29–36

[12] Olate S, Negreiros Lyrio M C, de Moraes M, Mazzonetto R and Fernandes Moreira R W 2010 Influence of diameter and length of implant on early dental implant failure *J. Oral. Maxil. Surg.* **68–2** 414–9

[13] Wu A Y J, Hsu J T, Chee W, Lin Y T, Fuh L J and Huang H L 2016 Biomechanical evaluation of one-piece and two-piece small-diameter dental implants: *in-vitro* experimental and three-dimensional finite element analysis *J. For. Med. Ass.* **115** 794–800

[14] Şimşek B, Erkmen E, Yilmaz D and Eser A 2006 Effects of different inter-implant distances on the stress distribution around endosseous implants in posterior mandible: a 3D finite element analysis *Med. Eng. & Phys.* **28** 199–213

[15] International Organization for Standardization (ISO) 2016ISO 14801: Dentistry—Fatigue test for endosseous dental implants (Accessed: https://www.iso.org/standard/61997.html)

[16] Shemtov-Yona K and Rittel D 2016 Fatigue of dental implants: facts and fallacies *Dent. J.* **4–2** 16–27

[17] Wu T, Fan H, Ma R, Chen H, Li Z and Yu H 2017 Effect of lubricant on the reliability of dental implant abutment screw joint: an *in vitro* laboratory and three-dimension finite element analysis *Mat. Sci. and Eng.* C **75** 297–304

[18] Kayabasi O, Yüzbasioglu E and Erzincanli F 2006 Static, dynamic and fatigue behaviors of dental implant using finite element method *Adv. Eng. Soft.* **37** 649–58

[19] Yang J and Xiang H J 2007 A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone *J. Biomech.* **40** 2377–85

[20] Wakabayashi N, Ona M, Suzuki T and Igarashi Y 2008 Nonlinear finite element analyses: advances and challenges in dental applications *J. Dent.* **36** 463–71

[21] Chun-Li L, Shih-Hao C, Jen-Chyan W and Wen-Jen C 2006 Mechanical interactions of an implant/tooth-supported system under different periodontal supports and number of splinted teeth with rigid and non-rigid connections *J. Dent.* **34** 682–91

[22] Cook R D 1995 *Introduction Finite Element Modeling for Stress Analysis* (New York: Wiley Inc.) chapter 1 pp 1–13

[23] National Institute of Industrial Technology (INTI-Madera y Muebles) 2017 (http://inti.gob.ar/maderaymuebles/pdf/caracterizacion_maderas/PINO_PARANA.pdf)

[24] Green D W, Winandy J E and Kretschmann D E 2010 Mechanical properties of wood *Wood Handbook: Wood as an Engineering Material* (Wisconsin: USDA Forest Service, Forest Products Laboratory) pp 5–2

[25] Djebbar N, Serier B, BachirBouiadjra B, Benbarek S and Drai A 2010 Analysis of the effect of load direction on the stress distribution in dental implant *Mater. and Design* **31** 2097–101

[26] Kong L, Gu Z, Hu K, Zhou H, Liu Y and Liu B 2009 Optimization of the implant diameter and length in type B/2 bone for improved biomechanical properties: a three-dimensional finite element analysis *Adv. Eng. Soft.* **40** 935–40

[27] Mesnard M and Ramos A 2012 Biomechanical characterization and modeling of natural and alloplastic human temporomandibular joint *Technologies for Medical Sciences, Lecture Notes in Computational Vision and Biomechanics* (Dordrecht: Springer) pp 39–66

[28] Geramy A, Adibrad M and Sahabi M 2010 The effects of splitting periodontally compromised removable partial denture abutments on bone stresses: a three-dimensional finite element study *J. Dent. S.* **5–1** 1–7

[29] Faegh S and Müftü S 2010 Load transfer along the bone–dental implant interface *J. Biomech.* **43** 1761–70

[30] Dieter G 1986 Fatigue of metals *Mechanical Metallurgy* (London: McGraw-Hill Book Co) pp 375–431

[31] Pazos L, Corengia P and Svoboda H 2010 Effect of surface treatments on the fatigue life of titanium for biomedical applications *J. Mech. Bio. Mater.* **3** 416–24

[32] Allum S R, Tomlinson R A and Joshi R 2008 The impact of loads on standard diameter, small diameter and mini implants: a comparative laboratory study *Clin. Oral. Impl. Res.* **19** 553–9