J Maxillofac Oral Surg 8(4):334–339

NEW ADVANCES

3D finite element analysis to detect stress distribution: spiral family implants

Matteo Danza[1] · **Ilaria Zollino**[2] · **Luigi Paracchini**[3] · **Guidi Riccardo**[2] · **Stefano Fanali**[1] · **Francesco Carinci**[2] ✉

[1] Dental School, University of Chieti, Italy
[2] Maxillofacial Surgery, University of Ferrara, Italy
[3] Private Practice, Milano, Italy

Received: 18 September 2008 / Accepted: 5 November 2009

Abstract
Aim Spiral family implants are a root-form fixtures with increasing thickness of tread. This characteristic gives a self-tapping and self-condensing bone properties to implants. To study spiral family implant inserted in different bone quality and connected with abutments of different angulations a Finite Element Analysis (FEA) was performed. Once drawn the systems that were object of the study by CAD (Computer Aided Design), the FEA discretized solids composing the system in many infinitesimal little elementary solids defined finite elements. This lead to a mesh formation where the single finite elements were connected among them by nodes. For the 3 units bone-implant-abutments several thousand of tetrahedral elements having 10 parabolic nodes were employed.
Materials and methods The biomechanical behaviour of 4.2 mm x 13 mm dental implants, connecting screw, straight and 15° and 25° angulated abutment subjected to static loads, in contact with high and poor bone quality was evaluated by FEA. A double system was analyzed: a) FY strength acting along Y axis and having 200 N intensity; b) FY and FZ couple of strengths applied along Y and Z directions and having respectively 200N and 140N intensity. The materials were considered as homogeneous, linear and isotropic. Then the FEA simulation was performed hypothesizing a linearity between loads and deformations.
Results The lowest stress value was found in the system composed by implants and straight abutments loaded with a vertical strength, while the highest stress value were found in implants and 15° angulated abutment loaded with a angulated strength. In addition, the lower is the bone quality (i.e. D4) the higher is the distribution of the stress within the bone.
Conclusion Spiral family implants can be used successfully in low bone quality but a straight force is recommended.

Keywords Biomechanics · Finite element analysis · Spiral · Implant · Stress · Distribution

Address for correspondence:

Francesco Carinci
Chair of Maxillofacial Surgery
Arcispedale S. Anna
Corso Giovecca 203
44100 Ferrara, Italy
Ph/Fax: +39.0532.455582
E-mail: crc@unife.it
Web: www.carinci.org

Introduction

The biomechanical behavior of an osseointegrated dental implant plays an important role in its functional longevity inside the bone.

Finite Element Analysis (FEA) has been used extensively to predict the biomechanical performance of various dental implant designs as well as the effect of clinical factors on implant success. By understanding the basic theory, method, application, and limitations of FEA in implant dentistry, the clinician will be better equipped to interpret results of FEA studies and extrapolate these results to clinical situations [1].

FEAs were performed for various shapes of dental implant to study effects on stress distribution generated in the surrounding jaw bone and to determine an optimal thread shape for stress distribution [2]. Once drawn the systems that were object of the study (for example an implant inserted in a mandible and bearing an abutment) by CAD (Computer Aided Design), the FEA discretized solids composing the system in many infinitesimal little elementary solids defined finite elements. This lead to a mesh formation where the single finite elements were connected among them by nodes. It is then possible to perform a simulation by applying forces to the system and detect the stress distribution to the single element.

It has been hypothesized that marginal bone resorption may result from micro damage accumulation in the bone. In light of this, a dental implant should be designed such that the peak stresses arising in the bone are minimized. The load on an implant can be divided into its vertical and horizontal components. In earlier studies, it was found that the peak bone stresses resulting from vertical load components and those resulting from horizontal load components arise at the top of the marginal bone, and that they coincide spatially. These peak stresses added together produce a risk of stress-induced bone resorption [3].

In addiction, the selection of the appropriate alignment between forces and implant are vital for its long-term success. Excessive load is generated around inclined implants, causing micro-cracks in the bone, which result in implant loosening and eventual failure [4].

Using FEA it was found that, with a conical implant-abutment interface at the level of the marginal bone, in combination with retention elements at the implant neck, and with suitable values of implant wall thickness and modulus of elasticity, the peak bone stresses resulting from an axial load arose further down in the bone. This meant that they were spatially separated from the peak stresses resulting from horizontal loads. If the same implant-abutment interface was located 2 mm more coronally, these benefits disappeared. This also resulted in substantially increased peak bone stresses [5]. As regard the abutment type, in vertical loading, stresses is concentrated around the implant-abutment connection at the stem of the screw and around the implant neck. Oblique loading resulted in a 2-fold increase in stresses at the implant neck, which was close to the yield strength of titanium [5].

Recently, a new type of implant with a spiral form has been produced (3D Alpha Bio - Pescara - Italy - Fig. 1). Spiral family implants are a root-form fixtures with increasing thickness of tread. This characteristic gives a self-tapping and self-condensing bone properties to implants. To study spiral family implant inserted in different bone quality and connected with abutments of different angulations and subjected to different forces a FEA was performed. Specifically, the FEA analyzes the stress distribution within two different bone types (i.e. high and poor qualities) due to forces applied to three implant' systems (i.e. one spiral implant plus one straight abutment, or one 15° angulated abutments, or one 25° angulated abutments). Specific aims were to detect the effect of spiral fixture on bone of different qualities and to detect the stress distribution within bone.

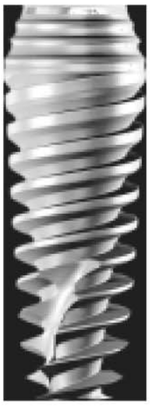


Fig. 1 A spiral family implant

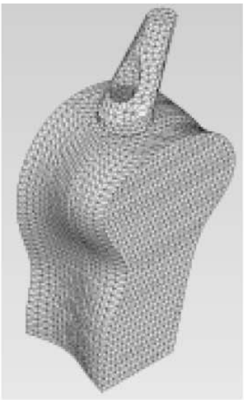
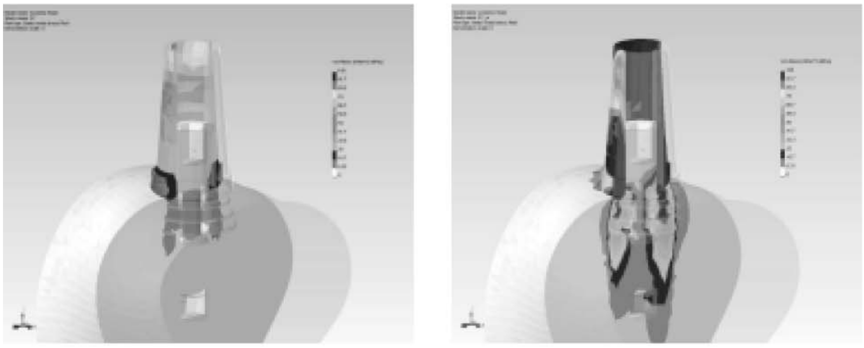
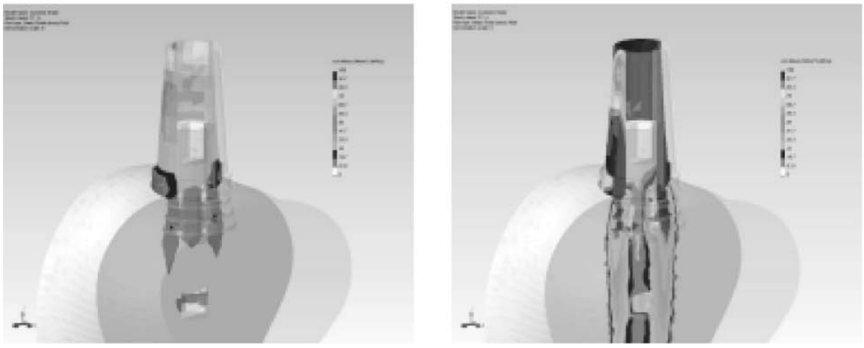


Fig. 2 Mesh formation where the single finite elements were connected among them by nodes



a. Bone quality D1 and force Fy = 200 N

b. Bone quality D1 and force Fy = 200 N plus Fz = 140N



c. Bone quality D4 and force Fy = 200 N

d. Bone quality D4 and force Fy = 200 N plus Fz = 140N

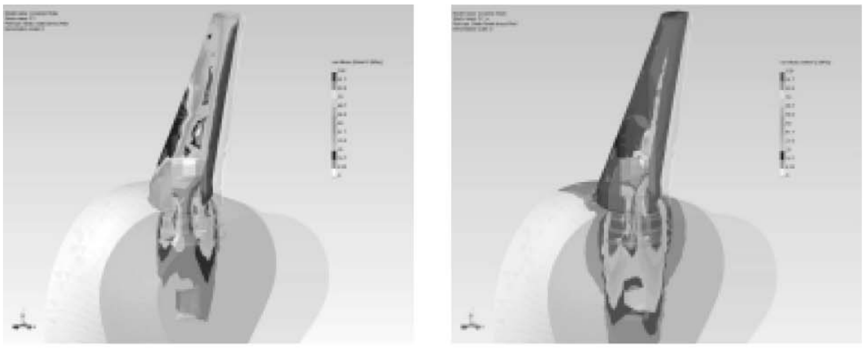
Fig. 3 Stress distribution (von_MISES) of SFB implant connected with straight abutment

Materials and methods

The biomechanical behaviour of an implant system (3D Alpha Bio – Pescara – Italy) subjected to static loading in contact with high (i.e. D1) and low (i.e. D4) bone tissue was evaluated in the present study. The implants were 4.2 mm in diameter and 13 mm in length and abutments were straight and 15° and 25° angulated. FEA was used in order to determine strain distribution in the tissues around the implant related to different bone structure, abutment angulations and loading. It was important to specify the implant system (i.e. implant plus abutment), the kind of bone, the entity of axial and transversal loads applied to the different configurations in order to evaluate the biomechanical behaviour. The directions of axial and transversal loads that stress implant and bone tissue when applied to the implant top were evaluated. A double system was analyzed: a) F_Y strength acting along Y axis and having 200N intensity; b) F_Y and F_Z couple of strengths applied along Y and Z directions and having respectively 200 N and 140N intensity. In order to plan the FEA and to reach the relative results it was necessary to create mathematical models that are curves, surfaces and solids. Once drawn the systems that were object of the study by Computer Aided Design (CAD), the FEA discretized solids composing the system in many infinitesimal little elementary solids defined finite elements. This lead to a mesh formation where the single finite elements were connected among them by nodes. For the 3 units bone-implant about 19000 nodes and about 105000 tetrahedral elements having 10 parabolic nodes were employed. Once the solids, the mesh and the planned loads (direction and intensity) were defined, a definition of the chemico-physical properties of materials was needed. For biomechanical analyses of materials subjected to low intensity strengths the materials can be considered homogeneous, linear and isotropic. Then the FEA simulation was performed hypothesizing linearity between loads and deformations. The portion of bone containing the implant was bound around two sides by joints removing all degrees of freedom to the system (Fig. 2).

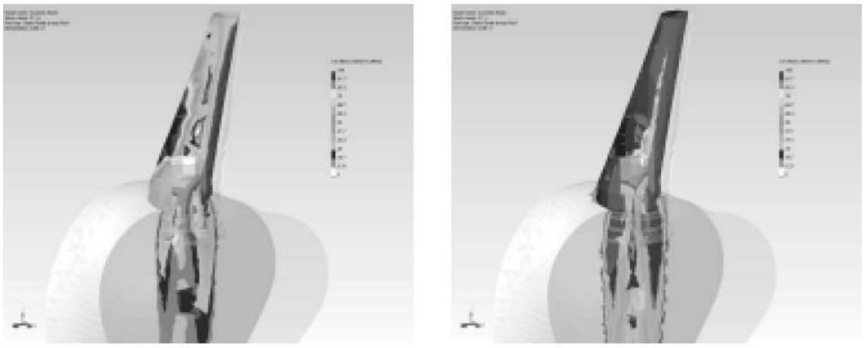
The pivot inside the bone tissue was bound by contact elements so as the connecting screw and the abutment. The CAD 3-D mathematical models used for FEA were realized using a surface modeller (RHINOCEROS 4.0 - McNeel Europe, Barcelona, Spain) and a solid modeller (SOLID WORKS 2007 SP2.2 - SolidWorks Corporation Headquarters, Concord, MA), both belonging to WINDOWS XP Professional Edition-SP1 - Microsoft Corporation, Milano, Italy). The discretization in finite elements and the FEA were realized by NEiFUSION 1.12 (Noran Engineering, Inc., Westminster, CA). The systems analyzed were the following (Figs. 3 to 5):

1. Implant having 4.2 mm diameter with 0° abutment: D1 bone and vertical strength;
2. Implant having 4.2 mm diameter with 0° abutment: D1 bone and tilted strength;
3. Implant having 4.2 mm diameter with 0° abutment: D4 bone and vertical strength;
4. Implant having 4.2 mm diameter with 0° abutment: D4 bone and tilted strength;
5. Implant having 4.2 mm diameter with 15° abutment: D1 bone and vertical strength;
6. Implant having 4.2 mm diameter with 15° abutment: D1 bone and tilted strength;



a. Bone quality D1 and force Fy = 200 N;

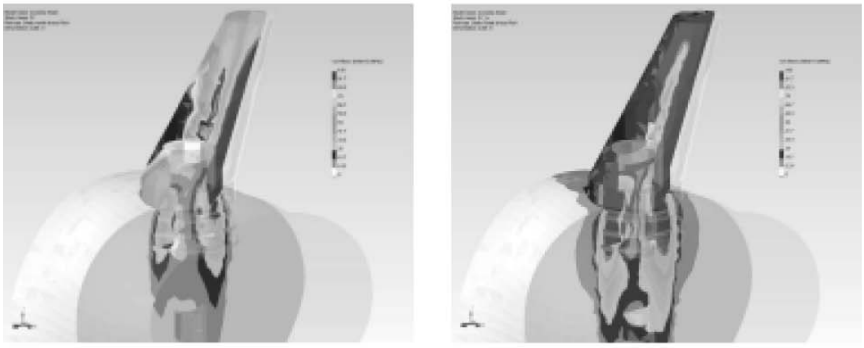
b. Bone quality D1 and force Fy = 200 N plus Fz = 140N



c. Bone quality D4 and force Fy = 200 N

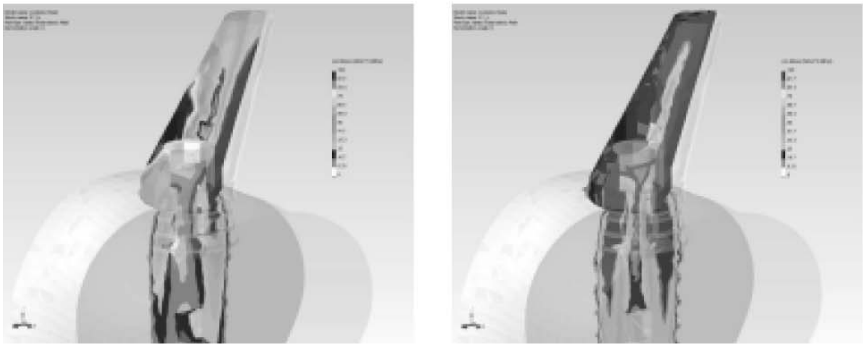
d. Bone quality D4 and force Fy = 200 N plus Fz = 140N

Fig. 4 Stress distribution (von_MISES) of SFB implant connected with 15 degrees abutment



a. Bone quality D1 and force Fy = 200 N;

b. Bone quality D1 and force Fy = 200 N plus Fz = 140N;



c. Bone quality D4 and force Fy = 200 N;

d. Bone quality D4 and force Fy = 200 N plus Fz = 140N

Fig. 5 Stress distribution (von_MISES) of SFB implant connected with 25 degrees abutment

Table 1

Particular	Material	“E” Young’s modulus (Pa)	“v” Poisson ratio (adimensional)
Mandible section	D1 Cortical bone	1.47E10 Pa	0.3
Mandible section	D4 Cortical bone	0.14E10 Pa	0.3
Fixture	Titanium	1.05E11 Pa	0.35
Connecting screw	Titanium	1.05E11 Pa	0.35
Abutment	Titanium	1.05E11 Pa	0.35

Table 2

	4.2 x 13 SFB Implant Straight Abu FY=200 N (σ_{MAX} Von Mises)	4.2 x 13 SFB Implant Straight Abu FY=200 N FZ = 140N (σ_{MAX} Von Mises)	4.2 x 13 SFB Implant 15° Abu FY=200 N (σ_{MAX} Von Mises)	4.2 x 13 SFB Implant 15° Abu FY=200 N FZ = 140 N (σ_{MAX} Von Mises)	4.2 x 13 SFB Implant 25° Abu FY=200 N (σ_{MAX} Von Mises)	4.2 x 13 SFB Implant 25° Abu FY=200 N FZ = 140 N (σ_{MAX} Von Mises)
D1	83	205	267.8	1025.5	246.3	671.8
D4	83	205	267.8	1025.5	246.3	671.8

Table 3

	4.2 x 13 SFB Implant Straight Abu FY=200 N ($\varepsilon_{MAX\ total\ micro\text{-}strain}$)	4.2 x 13 SFB Implant Straight Abu FY=200 N FZ = 140N ($\varepsilon_{MAX\ total\ micro\text{-}strain}$)	4.2 x 13 SFB Implant 15° Abu FY=200 N ($\varepsilon_{MAX\ total\ micro\text{-}strain}$)	4.2 x 13 SFB Implant 15° Abu FY=200 N FZ = 140 N ($\varepsilon_{MAX\ total\ micro\text{-}strain}$)	4.2 x 13 SFB Implant 25° Abu FY=200 N ($\varepsilon_{MAX\ total\ micro\text{-}strain}$)	4.2 x 13 SFB Implant 25° Abu FY=200 N FZ = 140 N ($\varepsilon_{MAX\ total\ micro\text{-}strain}$)
D1	0.00107	0.00988	0.0053	0.0154	0.00558	0.01372
D4	0.00274	0.00911	0.0048	0.0148	0.00517	0.01269

7. Implant having 4.2 mm diameter with 15° abutment: D4 bone and vertical strength;
8. Implant having 4.2 mm diameter with 15° abutment: D4 bone and tilted strength;
9. Implant having 4.2 mm diameter with 25° abutment: D1 bone and vertical strength;
10. Implant having 4.2 mm diameter with 25° abutment: D1 bone and tilted strength;
11. Implant having 4.2 mm diameter with 25° abutment: D4 bone and vertical strength;
12. Implant having 4.2 mm diameter with 25° abutment: D4 bone and tilted strength.

In Table 1 all the characteristic values of ‘E’ Young’s Modulus and ‘n’ Poisson Ratio have been reported [6–14].

Results

The results obtained with the FEA simulation showed the relationship between loads applied on the system, geometrical characteristics of materials, joints and strain. One of the most used theories for determining stress in bone matrix is by means of ‘von_MISES’ theory. This theory was applied to this experimentation in order to determine stress distribution at the bone-implant system interface. The figures of the single systems (Figs. 3–5) following the same nomenclature used in the list of materials and methods (section passing for X=0 through YZ plane) were presented. Color from yellow to red indicated stress. The beige color corresponded to zero or toward zero stress while red color indicated maximum stress. The total results of stress and strain were summarized in Tables 2–3.

Graphics 1 and 2 report the values of stress distribution in bone of high and low quality.

Discussion

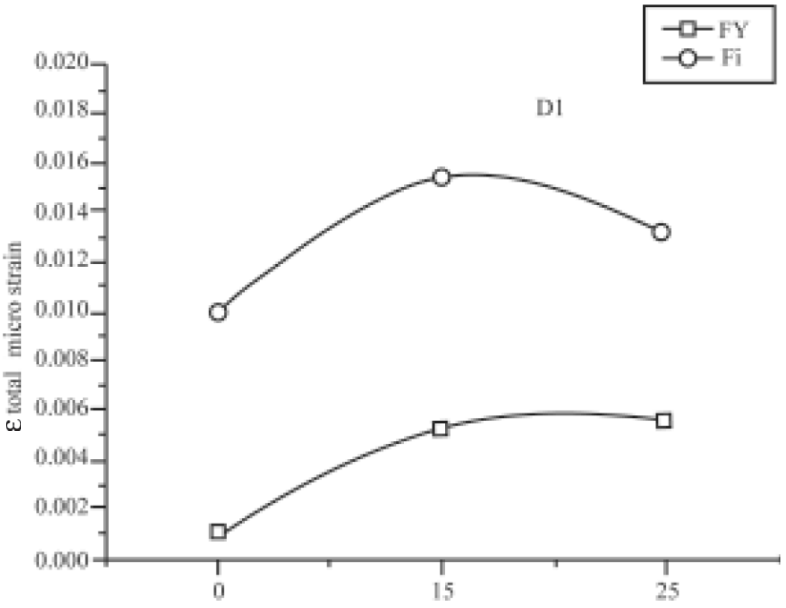
Primary implant stability and bone density are variables which are considered essential to achieve predictable osseointegration and long-term clinical survival of implants [15].

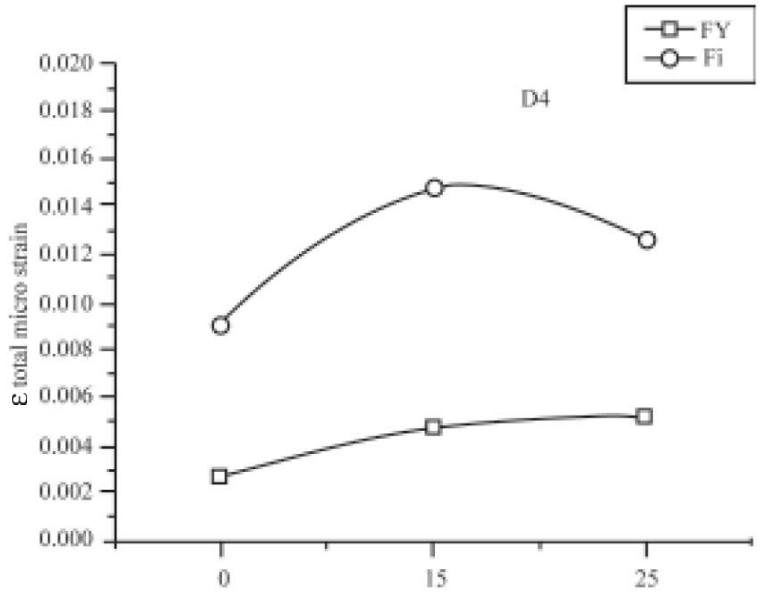
Zarb and Schmitt stated that bone structure is the most important factor in selecting the most favourable treatment outcome in implant dentistry [16]. Bone represents the external architecture of the edentulous area considered for implants. In addition, it has an internal structure described in term of quality or density, which reflects the strength of the bone [17].

For osseointegration of endosteal implants to occur, not only is adequate bone quantity required, but adequate density is also needed [18]. The initial bone density not only provides mechanical immobilization of the implant during healing, but after healing also permits distribution and transmission of stresses from the prosthesis to the implant bone interface [19].

The mechanical distribution of stress occurs primarily where bone is in contact with the implant [17]. Williams et al. [20] demonstrated that when maximum stress concentration occurs in cortical bone, it is located in the area of contact with the implant, and when the maximum stress concentration occurs in trabecular bone, it occurs around the apex of the implant. In cortical bone, stress dissipation is restricted to the immediate area surrounding the implant; in trabecular bone, a fairly broader distant stress distribution occurs [21].

Misch established that the percentage of bone contact is significantly greater in cortical bone than in trabecular bone [17]. Cortical bone, having a higher modulus of elasticity than trabecular bone, is stronger and more resistant to deformation [17]. For this reason, cortical bone will bear more load than trabecular bone in clinical situations [22].





The classification scheme for bone quality proposed by Lekholm and Zarb [23] has since been accepted by clinicians and investigators as standard in evaluating patients for implant placement. In this system, the sites are categorized into 1 of 4 groups on the basis of jawbone quality. In type 1 (D1) bone quality, the entire jaw is comprised of homogenous compact bone. In type 2 (D2) bone quality, a thick layer (2mm) of compact bone surrounds a core of dense trabecular bone. In type 3 (D3) bone quality, a thin layer (1mm) of cortical bone surrounds a core of dense trabecular bone of favorable strength. In type 4 (D4) bone quality, a thin layer (1mm) of cortical bone surrounds a core of low-density trabecular bone [17,23–26].

With the use of 3-dimensional FEA, Sevimay et al. [15] investigated the effect of these 4 different bone qualities on stress distribution in an implant-supported mandibular crown and they have been shown the presence of lower stresses for D1 and D2 bone quality and increased stresses for D3 and D4 bone quality because the trabecular bone was weaker and less resistant to deformation.

Bone is a porous material with complex micostructure. It is an anisotropic material, which means it has different physical properties when measured in different directions [1].

Canay et al. [27] compared vertically orientated implants with angled implants and found that the inclination of implants greatly influences stress concentrations around the implants.

In the present study a 3D FEA was performed to analyze the stress distribution within two different bone types (i.e. high and poor qualities) due to forces applied to three implant' systems (i.e. one spiral implant plus one straight abutment, or one 15° angulated abutments, or one 25° angulated abutments). The minimum bone stress was produced with straight abutment and vertical force whereas the maximum bone stress was obtained with 15° angulated abutment and coupled forces. In addition, the lower is the bone quality (i.e. D4) the higher is the distribution of the stress within the bone.

Conclusion

Spiral family implants can be used successfully in low bone quality but a straight force is recommended.

References

1. Geng JP, Tan KB, Liu GR (2001) Application of finite element analysis in implant dentistry: a review of the literature. J Prosthet Dent 85(6): 585–598
2. Chun HJ, Cheong SY, Han JH, Heo SJ, Chung JP, Rhyu IC, Choi YC, Baik HK, Ku Y, Kim MH (2002) Evaluation of design parameters of osseointegrated dental implants using finite element analysis. J Oral Rehabil 29(6): 565–574
3. Hansson S (2003) A conical implant-abutment interface at the level of the marginal bone improves the distribution of stresses in the supporting bone. An axisymmetric finite element analysis. Clin Oral Implants Res 14(3): 286–293
4. Watanabe F, Hata Y, Komatsu S, Ramos TC, Fukuda H (2003) Finite element analysis of the influence of implant inclination, loading position, and load direction on stress distribution. Odontology 91(1): 31–36
5. Akça K, Cehreli MC, Iplikçioðlu H. (2003) Evaluation of the mechanical characteristics of the implant-abutment complex of a reduced-diameter morse-taper implant. A nonlinear finite element stress analysis. Clin Oral Implants Res 14(4): 444–454
6. Natali AN, Pavan PG, Ruggero AL (2006) Analysis of bone-implant interaction phenomena by using a

numerical approach. Clin Oral Implants Res 17(1): 67–74
7. Akagawa Y, Sato Y, Teixeira ER, Shindoi N, Wadamoto M (2003) A mimic osseointegrated implant model for three-dimensional finite element analysis. J Oral Rehabil 30(1): 41–45
8. Iplikcioglu H, Akca K, Cehreli MC, Sahin S (2003) Comparison of non-linear finite element stress analysis with in vitro strain gauge measurements on a Morse taper implant. Int J Oral Maxillofac Implants 18(2): 258–265
9. Cruz M, Wassall T, Toledo EM, Barra LP, Lemonge AC (2003) Three-dimensional finite element stress analysis of a cuneiform-geometry implant. Int J Oral Maxillofac Implants 18(5): 675–684
10. Geng JP, Xu DW, Tan KB, Liu GR (2004) Finite element analysis of an osseointegrated stepped screw dental implant. J Oral Implantol 30(4): 223–233
11. Koca OL, Eskitascioglu G, Usumez A (2005) Three-dimensional finite-element analysis of functional stresses in different bone locations produced by implants placed in the maxillary posterior region of the sinus floor. J Prosthet Dent 93(1): 38–44
12. Satoh T, Maeda Y, Komiyama Y (2005) Biomechanical rationale for intentionally inclined implants in the posterior mandible using 3D finite element analysis. Int J Oral Maxillofac Implants 20(4): 533–539
13. Heckmann SM, Karl M, Wichmann MG, Winter W, Graef F, Taylor TD (2006) Loading of bone surrounding implants through three-unit fixed partial denture fixation: a finite-element analysis based on in vitro and in vivo strain measurements. Clin Oral Implants Res 17(3): 345–350
14. Kwon BG, Kim SG (2006) Finite Element Analysis of Different Bone Substitutes in the Bone Defects Around Dental Implants. Implant Dent 15(3): 254–264
15. Sevimay M, Turhan F, Kilicarslan MA, Eskitascioglu G (2005) Three-dimensional finite element analysis of the effect of different bone quality on stress distribution in an implant-supported crown. J Prosthet Dent 93(3): 227–234
16. Zarb GA, Schmitt A (1995) Implant prosthodontic treatment options for the edentulous patient. J Oral Rehabil 22(8): 661–671
17. Misch CE (1990) Density of bone: effect on treatment plans, surgical approach, healing, and progressive bone loading. Int J Oral Implantol 6(2): 23–31
18. Misch CE (1998) Contemporary implant dentistry. 2nd ed. St. Louis: Mosby 109–134, 207–217, 329–343, 595–608
19. Schroeder A (1996) Oral implantology: basic, ITI hollow cylinder system. New York: Thieme Medical Publishers 60–65
20. Williams KR, Watson CJ, Murphy WM, Scott J, Gregory M, Sinobad D (1990) Finite element analysis of fixed prostheses attached to osseointegrated implants. Quintessence Int 21(7): 563–570
21. Clift SE, Fisher J, Watson CJ (1992) Finite element stress and strain analysis of the bone surrounding a dental implant: effect of variations in bone modulus. Proc Inst Mech Eng H 206(4): 233–241
22. Cochran DL (2000) The scientific basis for and clinical experiences with Straumann implants including the ITI dental implant system: a consensus report. Clin Oral Implants Res. 11: 33–58
23. Lekholm U, Zarb GA (1985) Tissue-integrated prostheses. In: Branemark Pl, Zarb GA, Albrektsson T. Tissue-integrated prostheses. Chicago: Quintessence 199–209
24. Linkow LI, Rinaldi AW, Weiss WW Jr, Smith GH (1990) Factors influencing long-term implant success. J Prosthet Dent 63(1): 64–73
25. Bass SL, Triplett RG (1991) The effects of preoperative resorption and jaw anatomy on implant success. A report of 303 cases. Clin Oral Implants Res 2(4): 193–198
26. Hutton JE, Health MR, Chai JY, Harnett J, Jemt T, Johns RB, et al. (1995) Factors related to success and failure rates at 3-year follow-up in a multicenter study of overdentures supported by Branemark implants. Int J Oral Maxillofac Implants 10(1): 33–42
27. Canay S, Hersek N, Akpinar I, Asik Z (1996) Comparison of stress distribution around vertical and angled implants with finite-element analysis. Quintessence Int 27(9): 591–598

Source of Support: Nil, **Conflict of interest:** None declared.