中國機械工程學刊第四十三卷第五期第 463~471 頁(民國一百一十一年)
Journal of the Chinese Society of Mechanical Engineers, Vol.43, No.5, pp.463~471 (2022)

Improved Design for a Two-Piece Dental Implant System Using Fatigue and Torsion Testing Simulation

Danang Yudistiro*, Yung-Chang Cheng** and Cho-Pei Jiang***

Keywords： Dental implant, Fatigue safety factor, von Mises stress, ANSYS/Workbench, Uniform design, Desirability method.

ABSTRACT

This study improves a dental implant system using a fatigue and torsion testing simulation and a uniform design of experiment. ISO 14801 fatigue and ISO 13498 torsion testing standards for the fatigue safety factor and the von Mises stress for a two-piece implant model are determined using ANSYS/Workbench software. The two-piece implant model includes an implant and an abutment with a screw. The six control factors for a dental implant system are used to increase the fatigue safety factor and decrease the von Mises stress. All control factors are continuous in the design space so a uniform design is used to construct a group of simulation experiments. A suitable uniform table is used for the uniform design to increase the strength. After the uniform design of experiment, the improved design model is achieved using the desirability method. The improved design has a maximum increase in the fatigue safety factor of 9.6 % and the von Mises stress is reduced by 6%, compared to the original design. The strength of the implant is increased using a uniform design procedure.

INTRODUCTION

Paper Received May, 2022. Revised June, 2022, Accepted August, 2022, Author for Correspondence: Yung-Chang Cheng.

* *Graduate student, Ph. D. Program in Engineering Science and Technology, College of Engineering, National Kaohsiung University of Science and Technology, Taiwan 824, ROC.*

** *Corresponding author: Professor, Department of Mechatronics Engineering, National Kaohsiung University of Science and Technology, Kaohsiung, Taiwan 824, ROC.*

** *Professor, Department of Mechanical Engineering, National Taipei University of Technology, Taipei, Taiwan 106, ROC.*

Dental implants have been used as a substitute for teeth since ancient times. Osseointegration is a common subject for study and dental implants are used to replace and reconstruct decayed teeth (Topkaya et al., 2015). Dental implants are used as a substitute for teeth but they are not as strong as real teeth so the dental implant's structural strength and geometric design must meet the patient's requirements.

Previous studies determine the static and dynamic stress using the stress concentration, the displacement and osseointegration stability (Cheng et al., 2015; Dundar et al., 2016; Jiang et al., 2014; NarendraKumar et al., 2018; Paracchini et al., 2020). Finite element analysis was used by Paracchini et al. (2020) to determine the stress distribution in the cortical and cancellous bone that surrounds two models of dental implants. Using ANSYS software, NarendraKumar et al. (2018) determined the stress distribution and the deformation for four different implant designs using various thread angles. For two implant models with different stress geometry, Dundar et al. (2016) determined the stress distribution for three different external loadings using ANSYS software. Using ANSYS/LS-DYNA software, Cheng et al. (2015) and Jiang et al. (2014) calculated the micro-motion (displacement) of the cortical and cancellous bone under dynamic chewing loads.

The fatigue life and behavior of a dental implant model have been the subject of several studies (Geramizadeh et al., 2018; Kayabaşı et al., 2006; Liu et al., 2016; Prados-Privado et al., 2019). For a V-shaped thread, Geramizadeh et al. (2018) determined the fatigue behavior of dental implants using ANSYS software. Using the ISO 14801 testing standard, Prados-Privado et al. (2019) determined the fatigue limit and failure probability for titanium dental implants using ANSYS/Workbench software. Using the results of a study by Goodman, Soderberg and Gerber and the mean-stress fatigue criterion, the fatigue life and behavior of the Ti–6Al–4V implant was determined by Kayabaşı et al. (2006). Using

ANSYS/Workbench software, the biomechanical and fatigue safety factors for a titanium implant were determined by Liu et al. (2016).

Implant design affects osseointegration stability. The optimal design for a dental implant model has been the subject of previous studies (Elleuch, 2021; Gupta et al., 2021; Cheng et al., 2019). Using a desirability function and response surfaces, a study by Elleuch (2021) used a multi-objective optimization to reduce the equivalent stresses for an implant. Gupta et al. (2021) used topology optimization to optimize the design to decrease the von Mises stress and the axial deformation for a dental implant model. Using a uniform design, Kriging interpolation and a genetic algorithm, Cheng et al. (2019) optimized the design for a one-piece zirconia ceramic dental implant for dynamic chewing loads.

Previous studies show that the von Mises stress and the deformation for a dental implant model are the objective function for the design process. However, a torsion testing simulation for a dental implant model has never been used. A torsion testing simulation of a dental implant model is important for the surgical and chewing procedure. This study uses ISO 14801 fatigue and ISO 13498 torsion testing standards to determine the fatigue safety factor and von Mises stress for a dental implant model.

This study uses a uniform design of experiment to increase the structural strength of a dental implant system. A uniform design of experiment is used to generate a group of simulation experiments in the design space. A 3D model of a two-piece dental implant system that includes an abutment and implant, is constructed. For fatigue and torsion testing simulations, the minimum fatigue safety factor and the maximum von Mises stress for each implant model are calculated using ANSYS/Workbench software. A uniform design of experiment and the desirability method are used to improve the design for a two-piece dental implant system.

FINITE ELEMENT ANALYSIS USING SOFTWARE

Dental Implant Model

A 3-D model of a two-piece, tapered dental implants was drawn using SolidWorks CAD software. The dental implant models are 10 mm high and have a 25° taper angle, as shown in Figure 1. The two-piece dental implant system includes an implanted part and an abutment part (Kowalski et al., 2021). A two-piece dental implant produces less stress and strain on the peri-implant bone than a one-piece implant (Wu et al., 2016). A study by Duda et al. (2016) showed that a two-piece dental implant incurs less Marginal Bone Loss (MBL) than a one-piece dental implant, but there is no statistically significant difference between the two methods.

The special features of the dental implant system include a V-shaped thread on the implant body model and an abutment part, as shown in Figure 2. The V-shaped thread has a characteristic triangular shape with a pointed tip. The form of this thread is demonstrated by several studies to increase osseointegration stability. Geramizadeh et al. (2018) showed that a V-shaped design gives a uniform stress distribution around the cortical bone. This study uses six parameters to increase the strength of an implant: Implant Thread Depth (ITD), Implant Thread Pitch (ITP), Abutment Thread Depth (ATD), Abutment Thread Pitch (ATP), Abutment Body Size (ABS) and Abutment Thread Length (ATL), as shown in Figure 2. The dimensions for these six parameters are listed in Table 1.

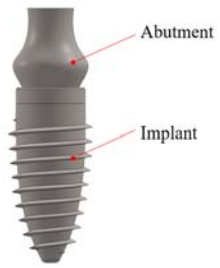


Fig. 1. 3D model of the dental implant.

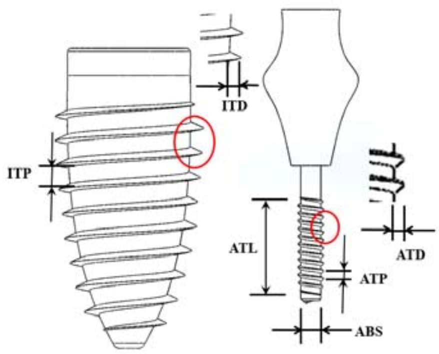


Fig. 2. Geometric dimensions of the implant.

Table 1. Geometric properties

ITD (mm)	ITP (mm)	ATD (mm)	ATP (mm)	ABS (mm)	ATL (mm)
0.45	0.75	0.4	0.75	1.25	5.5

Fatigue Finite Element Modeling

The standard for fatigue testing was developed by the Organization for International Standardization in 2003. In 2007, the ISO standard was increased to 5 million cycles (ISO 14801, 2013). Finite Elements Analysis (FEA) uses the ISO 14801 standard. The axial load that acts on the cap is 300 N. Figure 3 shows the arrangement for FEA for dental implants for ISO 14801 standard. The fatigue tests for this study use rigid and fixed clamping devices.

The performance index for the fatigue test for this study is the safety element. The safety factor is the ratio of the structure's load-bearing capacity to the predicted load. Static, dynamic and impact forces are examples of these loads. To prevent failure of the structural design, a safety factor is specified so the fatigue test safety factor is used to forecast mechanical failure for dental implants. For this study, a dental implant is structurally safe if it has a safety factor of more than 1.

For the fatigue testing simulation analysis, the mechanical properties of four parts are shown in Table 2: the implant, the abutment, the holder and the cap. The implant and the abutment are constructed using Ti6Al4V. Figure 4 shows the S-N curve for Ti6Al4V for the fatigue finite element analysis (Scherrer et al., 2011).

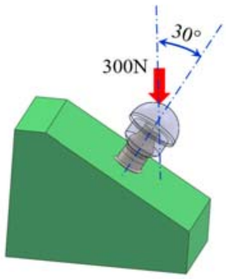


Fig. 3. Fatigue testing for the ISO 14801 standard

Table 2. Mechanical Properties of the dental implant testing system

Component	Density (kg/mm^3)	Young's modulus (MPa)	Poisson's ratio
Implant	4.5×10^{-6}	1.1×10^{5}	0.35
Abutment	4.5×10^{-6}	1.1×10^{5}	0.35
Cap	8×10^{-6}	1.93×10^{5}	0.25
Holder	4.5×10^{-6}	3.5×10^{3}	0.3
Specimen holder	7.85×10^{-6}	2×10^{5}	0.3

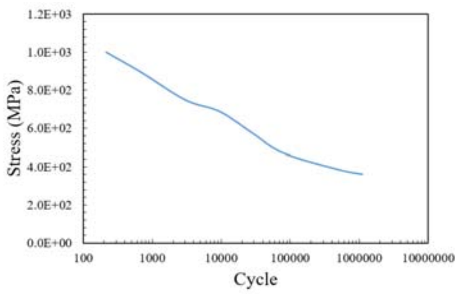


Fig. 4. The S-N curve for Ti6Al4V

The ISO-14801 testing standard assumes a boundary condition for the fatigue finite element analysis, as shown in Figure 5(a). An external loading is applied in the vertical direction and acts on the cap. The minimum fatigue safety factor is determined using ANSYS/Workbench software. Convergence analysis for the element meshing increases accuracy. Changes in the magnitude of the fatigue safety factor from 0.2-0.6 for various distinct elements are used to calculate the size of the elements. A change must be less than 5%.

Figure 5(b) shows the results for several different elemental sizes. A size of 0.3 mm is optimal because the difference in the simulation results for 0.2 and 0.3 is less than 5%. The fatigue safety factor for this ideal elemental size is shown in Figure 5(c). The minimum fatigue safety factor is 1.77, so a simulated endurance test shows that the dental implants are safe to use.

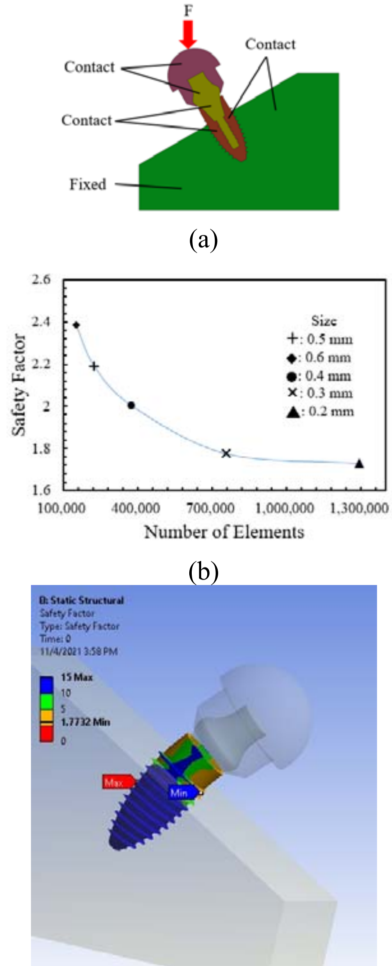


Fig. 5. (a) Boundary condition settings for FEA models, (b) the convergence analysis for elemental meshing and (c) the layout for the fatigue safety factor for the fatigue test simulation.

Torsion Finite Element Modeling

A torsion test procedure was developed for this study. Many dental implants use a dental implant body that is inserted into the jawbone and other components are then affixed to this body to construct a dental prosthetic. The connection between the components and the dental implant body must be strong and must withstand masticatory loads, including the torsional components.

ISO 13498 (2011) was developed in 2011 as a method to determine the torsional yield strength and the maximum torque on an implant body or connecting endosseous dental implants (ISO 13498, 2011). For this standard, the torsional yield strength and the maximum torque are determined by clamping the implant body and the connecting parts to be tested into the testing device. The implant body/connection is connected to the specimen holders using a maximum bond holder distance of 5mm. The static structural moment component is 1.533 N-m for one part and the fixed part does not move. Figure 6 shows a standard torque test using a torsion testing device.

The torsion test simulation for ISO 13498 uses boundary conditions, as shown in Figure 7(a). The horizontal torsion is applied to the specimen holder using a torsional driving device. In order to increase the accuracy of the finite element analysis results, mesh convergence analysis is used. Figure 7(b) shows the results for several different elemental sizes. The optional size is 0.3 mm because the difference in the simulation results for 0.2 and 0.3 is less than 5%. Figure 7(c) demonstrates the von Mises stress for this ideal element size. The dental implant system must withstand a maximum pressure of 220.61 MPa.

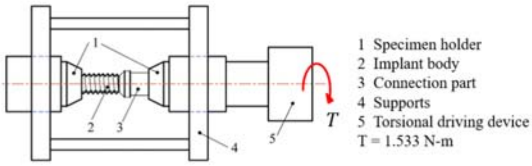
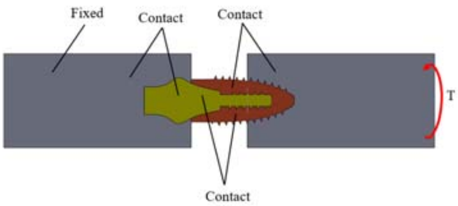
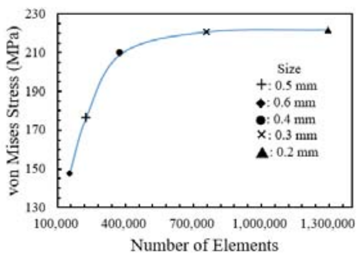


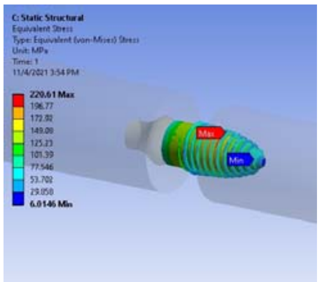
Fig. 6. Torsion testing for the ISO 13498 standard.



(a)



(b)



(c)

Fig. 7. (a) Boundary conditions for FEA models, (b) the convergence analysis for elemental meshing and (c) the von Mises stress for the torsion test simulation.

IMPROVED DESIGN FOR AN IMPLANT

Factor Analysis

This study uses a uniform design experimental method. This method reduces the number of simulations in order to reduce time and increase effectiveness and quality. The finite element analysis simulation is used to determine the value for the safety factor and the torque. The values for the simulation are then distributed. The uniformity of the value is used to determine the level of other factors in the simulation process. A uniform design gives more information than simulations or experiments using a smaller number of simulation samples.

The thread parameter is important to the primary stability and osseointegration of implants (Park et al., 2009). This study uses several key characteristics. Table 3 shows some of the key design features of dental implants. Six control factors are used for the dental implant system: ITD, ITP, ATD, ATP, ABS and ATL.

Table 3. Design ranges for the control factors

Control Factor	Lower bound	Basic Value	Upper Bound
ITD (mm)	0.3	0.45	0.6
ITP (mm)	0.5	0.75	1
ATD (mm)	0.3	0.4	0.5
ATP (mm)	0.3	0.75	1.2
ABS (mm)	1	1.25	1.5
ATL (mm)	4	5.5	7

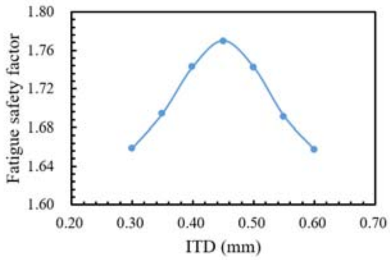
The effect of the implant thread depth (ITD) on the fatigue safety factor and von Mises stress is shown in Figures 8. In Figure 8(a), the fatigue safety factor increases and then decreases as the ITD is increased. In Figure 8(b), the von Mises stress increases as the thread depth in the implant increases.

Figures 9 show the effect of the implant thread pitch (ITP) on the fatigue safety factor and the von Mises stress. In Figure 9(a), the fatigue safety factor increases and then decreases as the thread pitch in the implant increases. The fatigue safety factor is higher in Figure 9(a). In Figure 9(b), the von Mises stress initially decreases and then increases as the ITP increases. The minimum von Mises stress is observed in Figure 9(b).

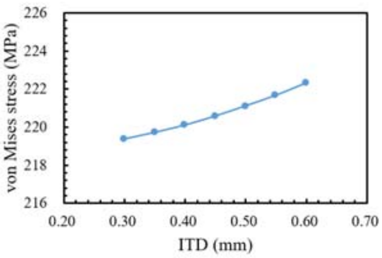
The effect of the abutment thread depth (ATD) on the fatigue safety factor and the von Mises stress is shown in Figures 10. In Figure 10(a), the fatigue safety factor initially decreases, then increases and finally decreases as the abutment thread depth increases. The local maximum fatigue safety factor is observed in Figure 10(a). In Figure 10(b), the von Mises stress increases as the ATD increases. This type of curve is also seen in Figure 8(b).

Figures 11 show the effect of the abutment thread pitch (ATP) on the fatigue safety factor and the von Mises stress. In Figure 11(a), the abutment thread pitch is demonstrated to have no effect on the fatigue safety factor. However, in Figure 11(b), the von Mises stress initially decreases and then increases as the ATP is increased. The minimum von Mises stress is observed in Figure 11(b).

Figures 12 show the effect of the abutment body size (ABS) on the fatigue safety factor and the von Mises stress. In Figures 12(a) and 12(b), the fatigue safety factor and the von Mises stress decrease very slightly as the size of the abutment body increases. Figures 13 show the effect of the abutment thread length (ATL) on the fatigue safety factor and the von Mises stress. In Figure 13(a), the fatigue safety factor increases significantly as the abutment thread length is increased. In Figure 13(b), the von Mises stress initially decreases and then increases as the ATL is increased. The type of curve in Figure 13(b) is also seen in Figures 8(b) and 10(b).

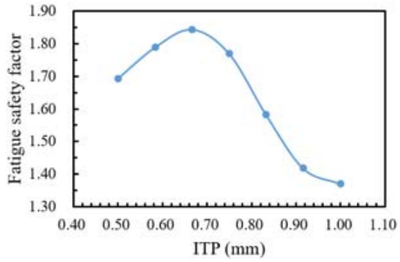


(a)

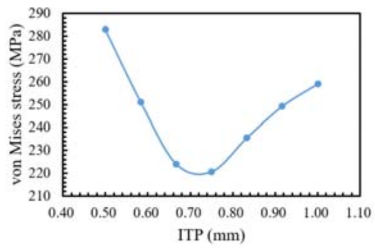


(b)

Fig. 8. The effect of the implant thread depth (ITD) on (a) the fatigue safety factor and (b) the von Mises stress.

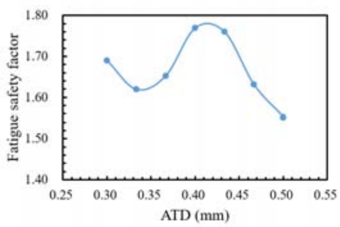


(a)

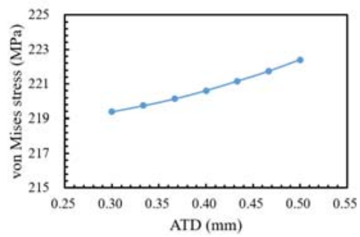


(b)

Fig. 9. The effect of the implant thread pitch (ITP) on (a) the fatigue safety factor and (b) the von Mises stress.

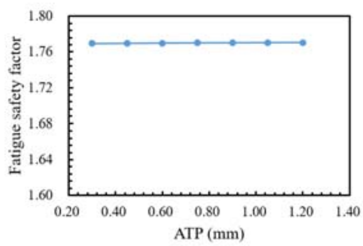


(a)

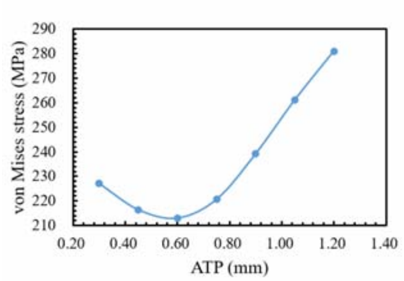


(b)

Fig. 10. The effect of the abutment thread depth (ATD) on (a) the fatigue safety factor and (b) the von Mises stress.

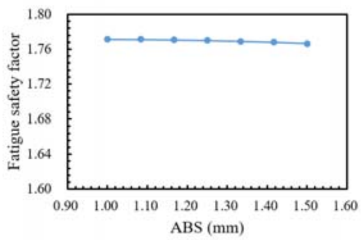


(a)

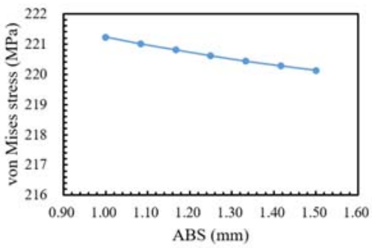


(b)

Fig. 11. The effect of the abutment thread pitch (ATP) on (a) the fatigue safety factor and (b) the von Mises stress.

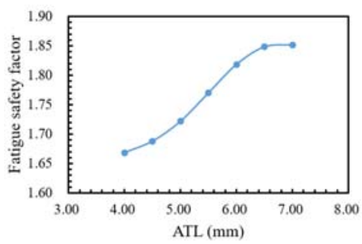


(a)

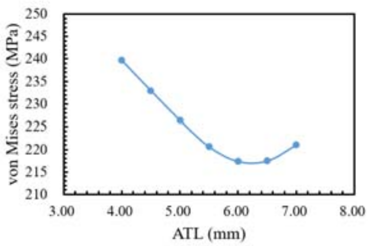


(b)

Fig. 12. The effect of the abutment body size (ABS) on (a) the fatigue safety factor and (b) the von Mises stress.



(a)



(b)

Fig. 13. The effect of the abutment thread length (ATL) on (a) the fatigue safety factor and (b) the von Mises stress.

Uniform Design of Experiment

The control factors are continuous so the design space can be regarded as a continuous space. Therefore, the uniform design of experiment method of Fang and Wang (2000) is used to construct a group of sample points that are dissipated evenly in an uninterrupted design space. This uniform design is widely used in many engineering areas. (Li et al., 2017; Li and Yang, 2019; Chatterjee et al., 2017; Lee et al., 2015)

If the number of experiments increases, the Kriging model that is established in the subsequent step is more accurate, but there is a significant increase in the computing time, so the number of experiments is limited. The minimum number of experiments is determined by the minimum number of input points that are required to initialize the Kriging model. Initializing the Kriging model requires at least 2n+1 design points, where n is the number of inputs. This study uses n inputs in this study so the minimum number of experiments is m=2n+1. Using a uniform design of experiment (Fang and Wang, 2000) and considering restrictions in machine instrumentation, each factor is assigned 16 levels, and 16 simulation tests are created using a uniform table $U_{16}^{*}\left(16^{12}\right)$.

A table of uniform design $U_{16}^{*}\left(16^{12}\right)$ (Fang and Wang, 2000) is used to create sixteen experiments, as shown in Table 4(a). This implant has six control factors - Implant Thread Depth, Implant Thread Pitch, Abutment Thread Depth, Abutment Thread Pitch, Abutment Body Size and Abutment Thread Length - so these control factors are used for the uniform design table. The results for sixteen tests are shown in Table 4. (b). Each experiment uses the SolidWorks geometric tool to generate a 3D solid dental implant model for a specific design of dental implant. Simulation tests use ANSYS/Workbench software. The fatigue safety factor and the von Mises stress values for all experiments are shown in Table 4(b).

In terms of the fatigue simulation, the safety factor for the original model is 1.77. In terms of the torsion test simulation, the von Mises stress is 220.61MP. In terms of improvement, experiments 3, 8, 9, 10 and 16 give a better safety factor. In terms of the von Mises stress, experiments 1, 2, 7 and 16 feature an improvement. The maximum fatigue safety factor for Y1 increases to 2.04 for the 9th experiment and the maximum von Mises stress for Y2 decreases to 203.48 MPa for the first experiment. These two results are not for the same experiment.

Table 4. (a) The experimental uniform design and (b) simulation results.

(a)

Exp. No.	ITD (mm)	ITP (mm)	ATD (mm)	ATP (mm)	ABS (mm)	ATL (mm)
1	0.3	0.6	0.37	0.84	1.43	6.80
2	0.32	0.73	0.45	0.42	1.33	6.40
3	0.34	0.87	0.30	1.02	1.23	6.00
4	0.36	1.00	0.38	0.6	1.13	5.60
5	0.38	0.57	0.46	1.2	1.03	5.20
6	0.40	0.70	0.31	0.78	1.50	4.80
7	0.42	0.83	0.39	0.36	1.40	4.40
8	0.44	0.97	0.47	0.96	1.30	4.00
9	0.46	0.53	0.33	0.54	1.20	7.00
10	0.48	0.67	0.41	1.14	1.10	6.60
11	0.50	0.80	0.49	0.72	1.00	6.20
12	0.52	0.93	0.34	0.30	1.47	5.80
13	0.54	0.50	0.42	0.90	1.37	5.40
14	0.56	0.63	0.50	0.48	1.27	5.00
15	0.58	0.77	0.35	1.08	1.17	4.60
16	0.6	0.9	0.43	0.66	1.07	4.20

(b)

Exp. No.	Safety Factor, Y1	von Mises stress, Y2 (MPa)
1	1.64	203.48
2	1.69	219.18
3	1.99	278.51
4	1.26	252.96
5	1.63	290.51
6	1.75	236.29
7	1.57	218.08
8	1.88	231.10
9	2.04	230.23
10	1.95	219.18
11	1.46	234.25
12	1.02	231.55
13	1.59	290.96
14	1.77	252.96
15	1.36	288.71
16	1.94	207.39

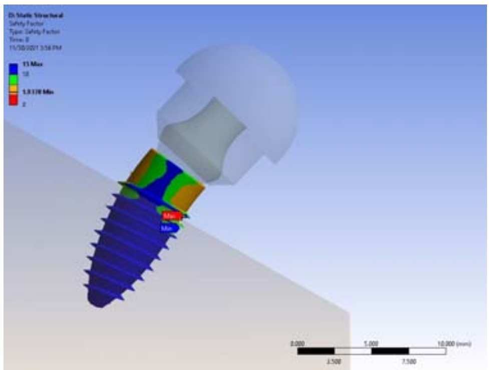
The desirability method is used to determine the desirability for the fatigue safety factor and the von Mises stress (Elleuch, 2021). The desirability for these two factors is shown in Table 5. The results in Table 5 show that the maximum desirability is derived in the 16th experiment, so the greatest improvement is observed for the 16th experiment. The analysis results for the 16th experiment are shown in Figure 14. For Y1 and Y2, the improvements are 9.6 % and 6.0 %, compared with the original design, as shown in Table 6. All of the objective values are improved.

Table 5. The desirability using the results of the uniform design.

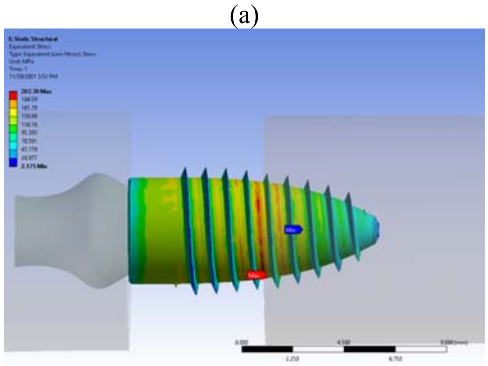
Exp. No.	d1	d2	Desirability	Rank
1	0.61	1.00	0.78	4
2	0.66	0.82	0.73	6
3	0.95	0.14	0.37	11
4	0.24	0.43	0.32	12
5	0.60	0.01	0.06	14
6	0.72	0.62	0.67	8
7	0.54	0.83	0.67	7
8	0.84	0.68	0.76	5
9	1.00	0.69	0.83	3
10	0.91	0.82	0.86	2
11	0.43	0.65	0.53	10
12	0.00	0.68	0.00	15
13	0.56	0.00	0.00	15
14	0.74	0.43	0.57	9
15	0.33	0.03	0.09	13
16	0.90	0.96	0.93	1

Table 6. Values and improvements in measures for various phases

Phase	Objective	Value	Improvement (%)
Original design	Fatigue safety factor, Y1	1.77	–
	von Mises stress, Y2 (MPa)	220.61	–
After uniform experiments	Fatigue safety factor, Y1	1.94	9.6
	von Mises stress, Y2 (MPa)	207.39	6.0



(a)



(b)

Fig. 14. The distribution of (a) the fatigue safety factor and (b) the von Mises stress for the improved design for fatigue and torsion testing simulations.

CONCLUSIONS

This study improves the design for a dental implant using a uniform design of experiment. ISO 14801 and 13498 testing standards specify that the fatigue safety factor and the von Mises stress are calculated using ANSYS/Workbench software. A convergence study using different elemental sizes is used to determine the quality of the meshing for the finite element analysis and the ideal elemental size for each FE model. Using a uniform design, there is an overall respective increase of 9.6 % in the fatigue safety factor and of 6% in the von Mises stress. The proposed improved design gives a dental implant system that is superior to the original design.

ACKNOWLEDGMENT

The authors gratefully acknowledge the financial support of the Ministry of Science and Technology (MOST) under grant no's, 107-2221-E-992-038-MY2 and 110-2221-E-992-069.

REFERENCES

Chatterjee K., Ou Z., Phoa F. K. H. and Qin H., "Uniform Four-Level Designs from Two-Level Designs: a New Look," *Statistica Sinica*, Vol. 27, pp. 171-186 (2017).

Cheng Y. C., Jiang C. P. and Lin D. H., "Finite element based optimization design for a one-piece zirconia ceramic dental implant under dynamic loading and fatigue life validation," *Structural and Multidisciplinary Optimization*, Vol. 59, No. 3, pp. 835–849 (2019).

Cheng Y. C., Lin D. H., Jiang C. P. and Lee S. Y., "Design improvement and dynamic finite element analysis of novel ITI dental implant under dynamic chewing loads," *Bio-Medical Materials and Engineering*, Vol. 26, No. s1, pp. S555–61 (2015).

Duda M., Matalon S., Lewinstein I., Harel N., Block J. and Ormianer Z., "One Piece Immediately Loaded Implants Versus 1 Piece or 2 Pieces Delayed: 3 Years Outcome," *Implant Dentistry*, Vol. 25, No. 1, pp. 109–113 (2016).

Dundar S., Topkaya T., Solmaz M. Y., Yaman F., Atalay Y., Saybak A., Asutay F. and Cakmak O., "Finite element analysis of the stress distributions in peri-implant bone in modified and standard-threaded dental implants," *Biotechnology & Biotechnological Equipment*, Vol. 30, No. 1, pp. 127–33 (2016).

Elleuch S., Jrad H., Kessentini A., Wali M. and Dammak F., "Design optimization of implant geometrical characteristics enhancing primary stability using FEA of stress distribution around dental prosthesis," *Computer Methods in Biomechanics and Biomedical Engineering*, Vol. 24, No. 9, pp. 1035–1051 (2021).

Fang K. T., Lin D. K. J., Winker P. and Zhang Y., "Uniform Design: Theory and Application," *Technometrics*, Vol. 42, No. 3, pp. 237–48 (2000).

Geramizadeh M., Katoozian H., Amid R. and Kadkhodazadeh M., "Comparison of finite element results with photoelastic stress analysis around dental implants with different threads," *Dental and Medical Problems*, Vol. 55, No. 1, pp. 17–22 (2018).

Gupta Y., Iyer R., Dommeti V. K., Nutu E., Rana M., Merdji A., Biswas J. K. and Roy S., "Design of Dental Implant using Design of Experiment and Topology Optimization: A Finite Element Analysis Study," *Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine*, Vol. 235, No. 2, pp. 157–166 (2021).

ISO14801. Fatigue Test for Endosseous Dental Implants. International Organization for Standardization; 2013.

ISO13498. Dentistry — Torsion test of implant body/connecting part joints of endosseous dental implant systems; 2011.

Jiang C. P., Lee C. K., Tsai W. L. and Cheng Y. C., "Uniform Design and Dynamic Finite Element Analysis for Micromotion Improvement of Semados Dental Implant System under Dynamic Chewing Loads," *Journal of Mechanics in Medicine and Biology*, Vol. 14, No. 6, pp. 1440007-1~144007-9 (2014).

Kayabaşı O., Yüzbasıoğlu E. and Erzincanlı F., "Static, dynamic and fatigue behaviors of dental implant using finite element method," *Advances in Engineering Software*, Vol. 37, No. 10, pp. 649–658 (2006).

Kowalski J., Lapinska B., Nissan J. and Lukomska-Szymanska M., "Factors Influencing Marginal Bone Loss around Dental Implants: A Narrative Review," *Coatings*, Vol. 11, No. 7, pp. 865-1~865-15 (2021).

Lee C. K., Cheng Y. C. and Jiang C. P., "Explicit Dynamic Finite Element Analysis and Uniform Design with Kriging Interpolation and Optimization to Improve an on-Road Bicycle Frame Undergoing Drop-Mass Impact Test," *Journal of the Chinese Society of Mechanical Engineers*, Vol. 36, No. 4, pp. 353-361 (2015).

Li B., Wang W., Zhang X., Zhang D., Mu W. and Liu F., "Integrating Uniform Design and Response Surface Methodology to Optimize Thiacloprid Suspension," *Scientific Reports*, Vol. 7, No. 1,pp. 1-9 (2017).

Li T. Z. and Yang X. L., "An Efficient Uniform Design for Kriging-Based Response Surface Method and its Application," *Computers and Geotechnics*, Vol. 109, pp. 12-22 (2019).

Liu L., Zhang X., Zhou Y. F., Chen X. S. and Wang Y. L., "Influence on Fatigue and Biomechanics of Cone Fit of Dental Implant around the Surrounding Bone Tissue," *Materials Science Forum*, Vol. 872, pp. 281–286 (2016).

NarendraKumar U., Mathew A. T., Iyer N., Rahman F. and Manjubala I., "A 3D Finite Element Analysis of Dental Implants with Varying Thread Angles," *Materials Today: Proceedings*, Vol. 5, No. 5, pp. 11900–11905 (2018).

Paracchini L., Barbieri C., Redaelli M., Di Croce D., Vincenzi C. and Guarnieri R., "Finite Element Analysis of a New Dental Implant Design Optimized for the Desirable Stress Distribution in the Surrounding Bone Region," *Prosthesis*, Vol. 2, No. 3, pp. 225–236 (2020).

Park J. H., Lim Y. J., Kim M. J. and Kwon H. B., "The effect of various thread designs on the initial stability of taper implants," *The Journal of Advanced Prosthodontics*, Vol. 1, No. 1, pp. 19-25 (2009).

Prados-Privado M., Ivorra C., Martínez-Martínez C., Gehrke S. A., Calvo-Guirado J. L. and Prados-Frutos J. C., "A Finite Element Analysis of the Fatigue Behavior and Risk of Failure of Immediate Provisional Implants," *Metals*, Vol. 9, No. 5, pp. 535-1~535-9 (2019).

Scherrer S. S., Cattani-Lorente M., Vittecoq E., de Mestral F., Griggs J.A. and Wiskott H. W. A., "Fatigue behavior in water of Y-TZP zirconia ceramics after abrasion with 30μm silica-coated alumina particles," *Dent Mater*, Vol. 27, No. 2, pp. e28–42 (2011).

Topkaya T., Solmaz M. Y., Dündar S. and Eltas A., "Numerical Analysis of the Effect of Implant Geometry to Stress Distributions of Dental Implant System," *Cumhuriyet Dental Journal*, Vol. 18, No. 1, pp.17-24 (2015).

Wu A. Y. J., Hsu J. T., Chee W., Lin Y. T., Fuh L. J. and Huang H. L., "Biomechanical evaluation of one-piece and two-piece small-diameter dental implants: In-vitro experimental and three-dimensional finite element analyses," *Journal of the Formosan Medical Association*, Vol. 115, No. 9, pp. 794–800 (2016).

具疲勞與扭轉測試模擬之兩件式牙根系統改善設計

余達南　　鄭永長
國立高雄科技大學機電工程系

江卓培
國立台北科技大學機械工程系

摘 要

本研究利用疲勞和扭轉測試模擬，以及均勻實驗設計進行人工牙根系統的改善設計。依據 ISO 14801 疲勞和 ISO 13498 扭轉測試標準，使用 ANSYS/Workbench 軟體計算與分析兩件式牙根系統模型的疲勞安全係數和 von Mises 應力。兩件式牙根系統模型包括牙根本體和具螺紋的牙冠。牙根系統的六個控制因素用於增加疲勞安全係數並降低 von Mises 應力。因為所有控制因素在設計空間中都是連續的，因此，使用均勻實驗設計來構建一組模擬實驗。在進行均勻實驗設計後，利用期望函數方法，得到改善設計的方案。改善後的設計與原設計相比，疲勞安全係數最大提高了 9.6%，von Mises 應力降低了 6%。根據結果可知，使用均勻實驗設計有效地增加了牙根系統的強度。