Research Article

Stress and Displacement Analysis of Dental Implant Threads Using Three-Dimensional Finite Element Analysis

Aswin Yodrux*
Department of Materials Handling and Logistics for Engineering, Faculty of Engineering, King Mongkut's University of Technology North Bangkok, Bangkok, Thailand

Nantakrit Yodpijit
Center for Innovation in Human Factors Engineering and Ergonomics, Department of Industrial Engineering, Faculty of Engineering, King Mongkut's University of Technology North Bangkok, Bangkok, Thailand

Manutchanok Jongprasithpornt
Department of Industrial Engineering, Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok, Thailand

* Corresponding author. E-mail: aswin.y@eng.kmutnb.ac.th DOI: 10.14416/j.ijast.2018.09.002
Received: 3 August 2017; Accepted: 15 November 2017; Published online: 6 September 2018

Abstract

Dental implant is one of the most common ways to replace a missing tooth. The objective of this research project is to analyze the compressive stress, shear stress, and displacement of prosthetic tooth. This research is conducted using a three-dimensional finite element method to analyze biomechanical characteristics on dental implant systems at four different areas: abutment, implant, cortical bone, and cancellous bone. This research focuses on a variety of patents of dental implant threads from the United States Patent and Trademark Office. Findings reveal a new conceptual design of dental implant thread model. It is found that the maximum stress concentration is found at cortical bone but is still lower than the yield strength of materials. The new design of dental implant tread provides the minimum compressive stress, shear stress, and displacement for dental implant systems.

Keywords: Biomechanical analysis, Dental implant systems, Three-dimensional finite element method

1 Introduction

Dental implants have been extensively used during the last few decades as replacements for missing natural teeth [1]. Titanium metals has become the preferred materials for dental implants owing to their good biocompatibility, excellent corrosion resistance and suitable mechanical properties [2].

Three-dimensional finite element analysis is a useful tool that has been applied to simulate behavior biomechanical characteristics on dental implant systems under comprassive loading [3].

This research focuses on a variety of patents of dental implant threads from the United States Patent and Trademark Office to new design model of implant prosthetics [4]–[6]. The result of comprassive stress, shear stress and displacements have been studied.

2 Material and Methods

2.1 *Three-dimension model design*

Three-dimensional geometrical model of dental implant prosthetics and the surrounding bone was created by

Please cite this article as: A. Yodrux, N. Yodpijit, and M. Jongprasithpornt , "Stress and displacement analysis of dental implant threads using three-dimensional finite element analysis," *Applied Science and Engineering Progress*, vol. 12, no. 3, pp. 216–222, Jul.–Sep. 2019.

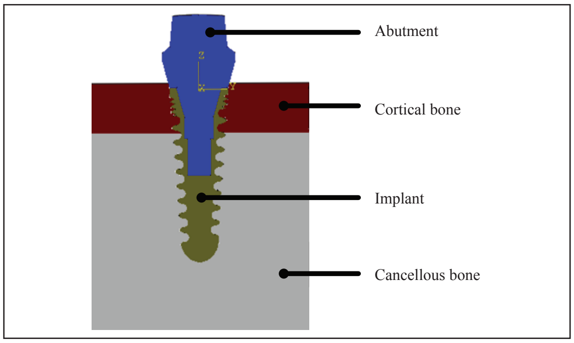


Figure 1: Three-dimensional geometrical model of dental implant prosthetic and the surrounding bone.

the Computer Aided Design (CAD) modelling for testing by three-dimensional finite element analysis. All the CAD models in this study are made, using CATIA program by solid modelling (Figure 1).

For the three-dimensional finite element analysis models generation, the implant thread was placed in a simplified jaw segment, a cancellous bone core surrounded by a 1.5 mm thick cortical bone layer was designed around the implant. The dimensions of the simulated bone block were 24.2 mm high and 16.3 mm wide [7], [8].

This research focuses on six implant prosthetics model of dental implant systems, from Figure 2 illustrated 3D thread category of implant prosthetics.

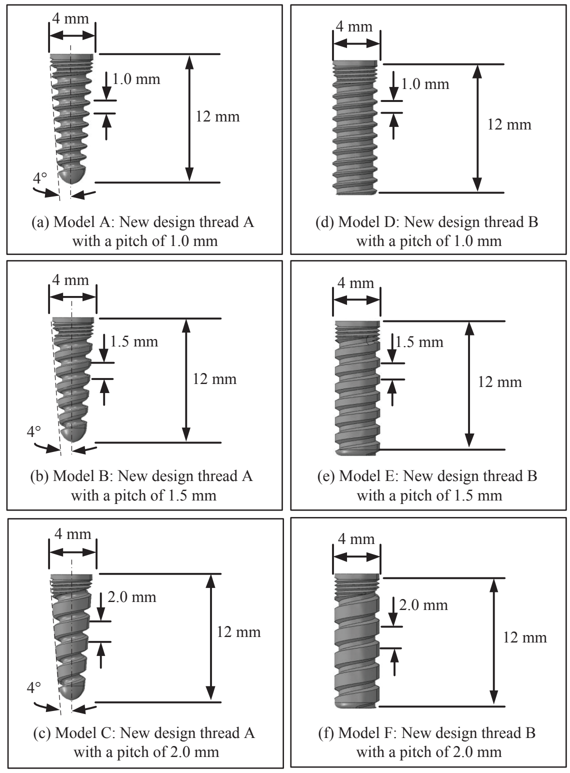


Figure 2: 3D thread category of six implant prosthetics model.

2.2 *Materials properties*

In the study, Titanium (Ti6Al-4V) was used as implant and abutment materials. The mechanical properties of materials were taken from the literature, as shown in Table 1 [9]–[14].

Table 1: Mechanical properties of materials

Materials	Young's modulus, E (MPa)	Poisson's Ratio, υ
Cortical Bone	13,000	0.3
Cancellous Bone	1,370	0.3
Implant (Ti6Al-4V)	102,000	0.35
Abutment (Ti6Al-4V)	102,000	0.35

2.3 *Loading and boundary condition*

The dental implant system is treated as isotropic homogeneous elastic materials. All material of the dental implant was assumed to be isotropic and linearly elastics.

Three-dimensional finite element analysis was performed on dental implant models with compressive forces of 100 N, and a shear force of 50 N with the force angle of 45° with the normal line respectively [15], [16]. Figure 3 show Loading conditions applied on dental implant system model.

The shape of the three-dimensional finite element analysis model of the dental implant prosthetics model is shown in Figure 4.

After loading, the result of comprassive stress, shear stress and displacements were calculated by three-dimensional finite element analysis for each of the different regions of the dental implant prosthetics model: abutment, implant, cortical bone, and cancellous bone.

3 Results

The compressive stress, shear stress and displacement

Applied Science and Engineering Progress, Vol. 12, No. 3, pp. 216–222, 2019

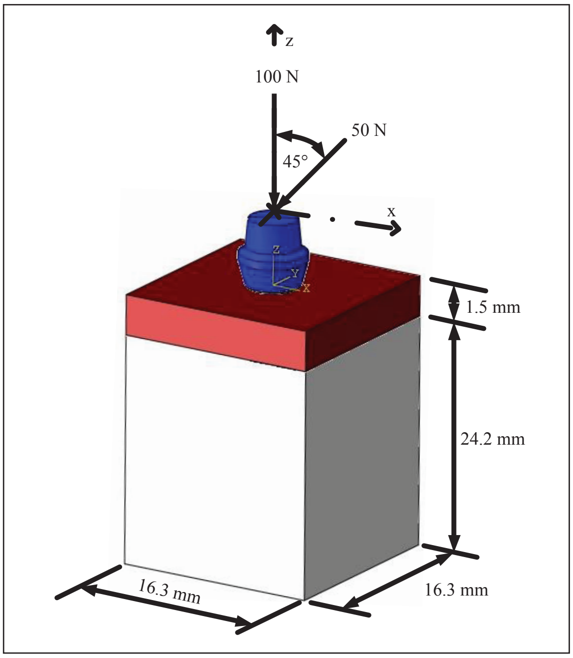


Figure 3: Loading conditions applied on dental implant system model.

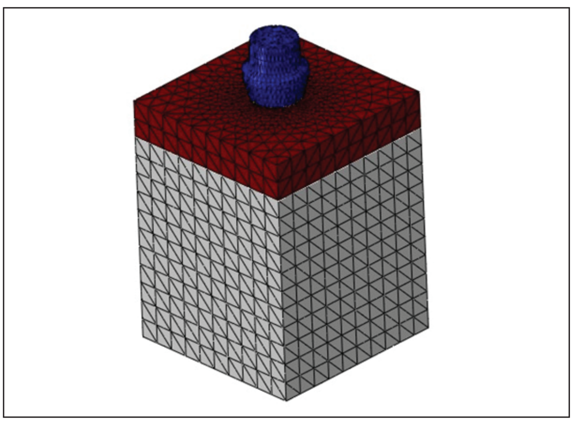
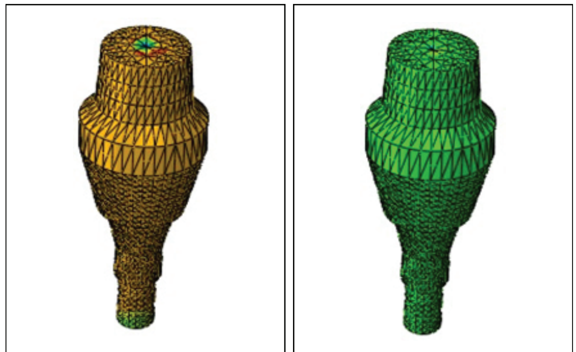


Figure 4: The three-dimensional finite element analysis model of the dental implant systems.

analysis were conducted at four different areas, including abutment, implant, cortical bone, and cancellous bone.

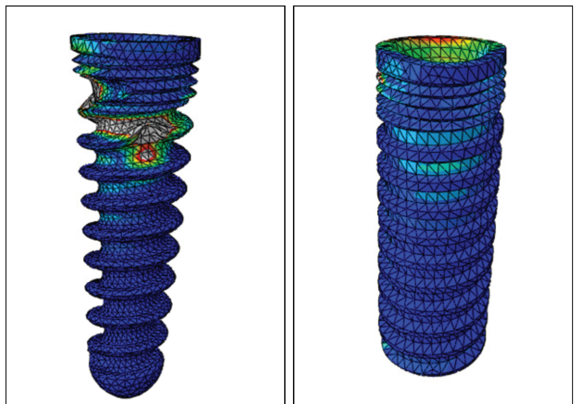
3.1 *Stress distribution*

As show in Figure 5, the resulting stress distribution in three-dimensional finite element analysis of abutments of dental implant prosthetics under compressive force.



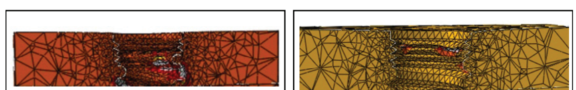
(a) New design thread A (b) New design thread B

Figure 5: The stress distribution of abutments.



(a) New design thread A (b) New design thread B

Figure 6: The stress distribution of implants.



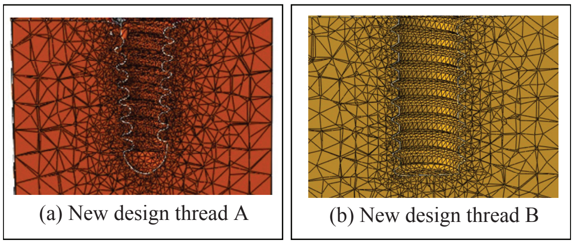
(a) New design thread A (b) New design thread B

Figure 7: The stress distribution of cortical bone.

Figure 6 illustrates the stress distribution in three-dimensional finite element analysis of implants of dental implant prosthetics under compressive force.

Figure 7 illustrates the stress distribution in three-dimensional finite element analysis of cortical bone under compressive force.

Figure 8 illustrates the stress distribution in three-dimensional finite element analysis of cancellous bone under compressive force.



(a) New design thread A (b) New design thread B

Figure 8: The principal stress of cancellous bone.

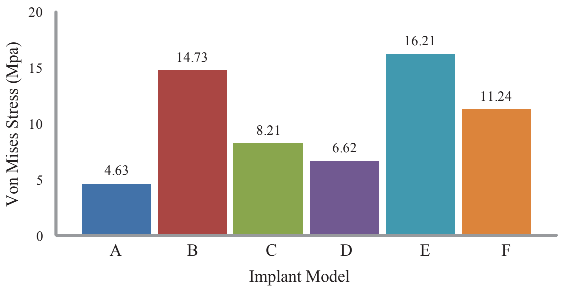


Figure 9: The Von Mises stress of abutments.

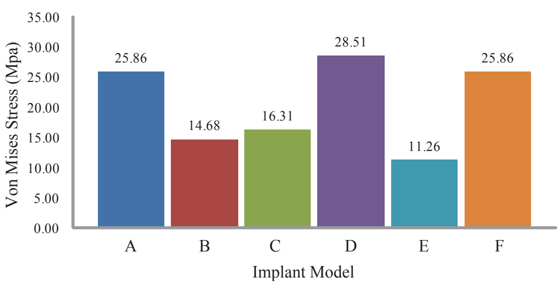


Figure 10: The Von Mises stress of implants.

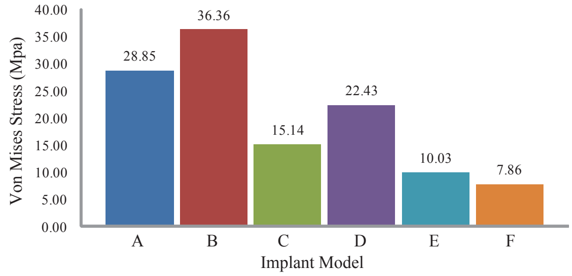


Figure 11: The Von Mises stress of cortical bone.

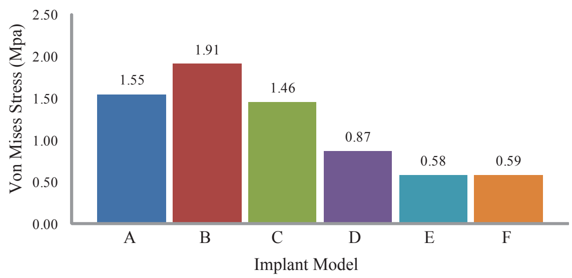


Figure 12: The Von Mises stress of cancellous bone.

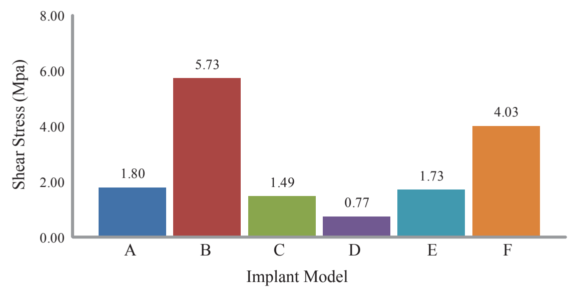


Figure 13: Shear stress of abutments.

3.2 *Von mises stress*

Results of the Von mises stress in three-dimensional finite element analysis of abutments of dental implant prosthetics under a compressive force are given in Figure 9. It reveals that the maximum stress of 16.21 MPa is found at thread E and the minimum stress of 4.63 MPa is found at thread A.

Results of the stress in three-dimensional finite element analysis of implants of dental implant prosthetics under a compressive force are given in Figure 10. It reveals that the maximum stress of 28.51 MPa is found at thread D and the minimum stress of 11.26 MPa is found at thread E.

Results of the stress in three-dimensional finite element analysis of cortical bone under a compressive force are given in Figure 11. It reveals that the maximum stress of 36.36 MPa is found at thread B and the minimum stress of 7.86 MPa is found at thread F.

Results of the stress in three-dimensional finite element analysis of cancellous bone under a compressive force are given in Figure 12. It reveals that the maximum stress of 1.91 MPa is found at thread B and the minimum stress of 0.58 MPa is found at thread E.

3.3 *Shear stress*

Results of shear stress in 3D-FEA of abutments under a shear force are given in Figure 13. It reveals that the maximum shear stress of 5.73 MPa is found at thread B and the minimum shear stress of 0.77 MPa is found

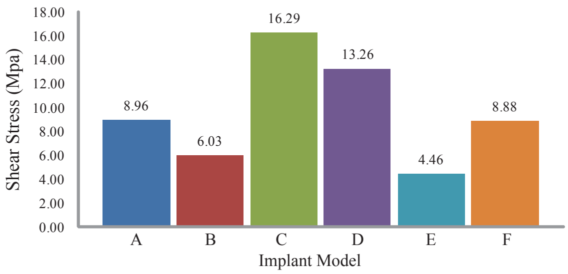


Figure 14: Shear stress of implants.

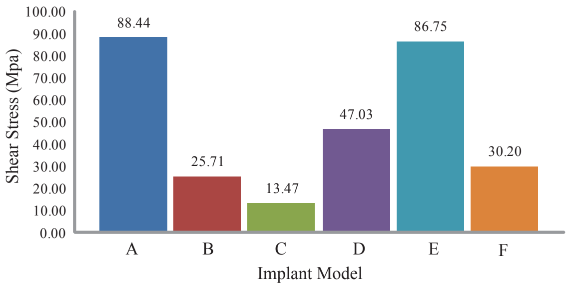


Figure 15: Shear stress of cortical bone.

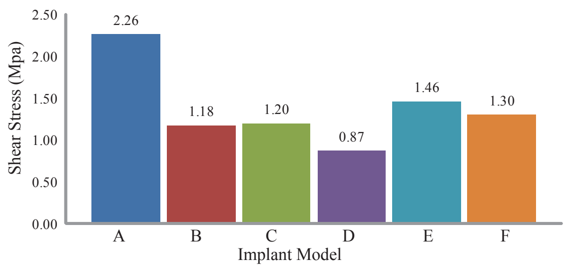


Figure 16: Shear stress of cancellous bone.

at thread D.

Results of shear stress in three-dimensional finite element analysis of implants under a shear force are given in Figure 14. It reveals that the maximum shear stress of 16.29 MPa is found at thread C and the minimum shear stress of 4.46 MPa is found at thread E.

Results of shear stress in three-dimensional finite element analysis of cortical bone under a shear force are given in Figure 15. It reveals that the maximum shear stress of 88.44 MPa is found at thread A and the minimum shear stress of 13.47 MPa is found at thread C.

Results of shear stress in three-dimensional finite element analysis of cancellous bone under a shear force are given in Figure 16. It reveals that the maximum shear stress of 2.26 MPa is found at thread A and the minimum shear stress of 0.87 MPa is found at thread D.

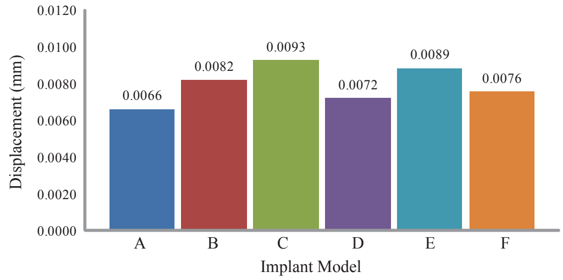


Figure 17: Displacement of abutments.

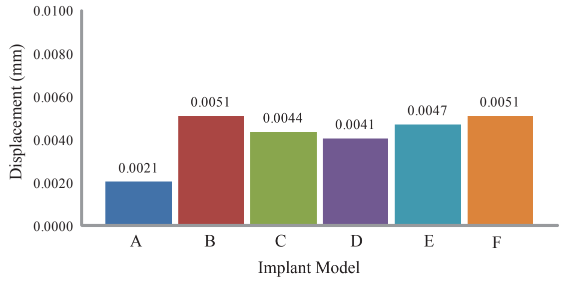


Figure 18: Displacement of implants.

3.4 *Displacement*

Results of displacement in three-dimensional finite element analysis of abutments under compressive force are given in Figure 17. It reveals that the maximum displacement of 0.0093 mm is found at thread C and the minimum displacement of 0.0066 mm is found at thread A.

Results of displacement in three-dimensional finite element analysis of implants under compressive force are given in Figure 18. It reveals that the maximum displacement of 0.0051 mm is found at thread B, F and the minimum displacement of 0.0021 mm is found at thread A.

Results of displacement in three-dimensional finite element analysis of cortical bone under compressive force are given in Figure 19. It reveals that the maximum displacement of 0.0078 mm is found at thread B, E and the minimum displacement of 0.0007 mm is found at thread D.

Results of displacement in three-dimensional finite element analysis of abutments under compressive force are given in Figure 20. It reveals that the maximum displacement of 0.0092 mm is found at thread C and the minimum displacement of 0.0009 mm is found at thread D.

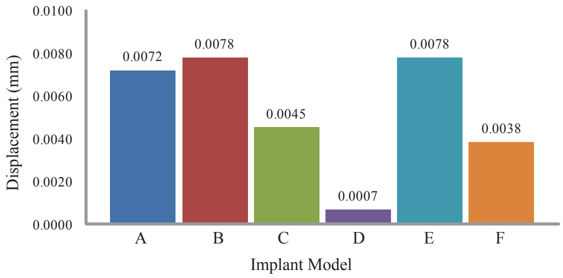


Figure 19: Displacement of cortical bone.

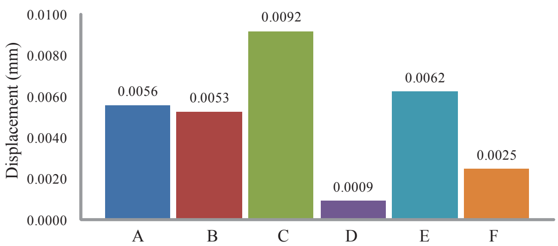


Figure 20: Displacement of cancellous bone.

4 Disccussion and Conclusions

The purpose of this research study was to analyze the compressive stress, shear stress, and displacement of prosthetic tooth. This research is conducted using a three-dimensional finite element method to analyze biomechanical characteristics on dental implant systems at four different areas: abutment, implant, cortical bone, and cancellous bone. In the study, Titanium (Ti6Al-4V) was used as implant and abutment materials. It was considered as the yield strength of Titanium grade 4 a value of 550 MPa. This resarch have focused on six different threads shape of dental prosthetics model. From this research, The maximum compressive stress occurred in four different areas, including abutment, implant, cortical bone, and cancellous bone are less than the yield strength of materials. The study of minimum shear stress and minimum displacement analysis occurred in four different areas, including abutment, implant, cortical bone, and cancellous bone are found at new design thread B with a picth of 1 mm. Hence the design is safe.

Acknowledgments

Authors would like to express our sincere appreciation to researchers and staff at the Center for Innovation in Human Factors Engineering and Ergonomics, Department of Industrial Engineering, Faculty of Engineering, King Mongkut's University of Technology North Bangkok (KMUTNB) for technical supports.

References

[1] Y. Gao, Y.-F. Li, B. Shao, T. Li, N. Xia, L.-X. Xu, Z.-Y. Wang, and L. Kong, "Biomechanical optimisation of the length ratio of the two endosseous portions in distraction implants: A three-dimensional finite element analysis," *British Journal of Oral and Maxillofacial Surgery*, vol. 50, pp. e86–e92, 2012.

[2] C. Liang-jian, H. Hao, L. Yi-min, L. Ting, G. Xiao-ping, and W. Rui-fang, "Finite element analysis of stress at implant-bone interface of dental implants with different structures," *Transactions of Nonferrous Metals Society of China*, vol. 21, pp. 1602–1610, 2011.

[3] L. Zheng, J. Yong, X. Hu, and J. Luo, "The dimensional finite element analysis of a novel osteointegrated dental implant designed to reduce stress peak of cortical bone," *Acta of Bioengineering and Biomechanics*, vol. 16, no. 3, pp. 21–28, 2014.

[4] H. Zipprich, "Dental Implant System," U.S. Patent 20140212844A1, Jul. 31, 2014.

[5] B.-W. Peng, "Dental Implant," U.S. Patent 0140816800A1, July 3, 2014.

[6] T. Buurlage, "Dental Implant," U.S. Patent 20140147808A1, May 29, 2014.

[7] C.-L. Lin, Y.-C. Kuo, and T.-S. Lin, "Effects of dental implant length and bone quality on biomechanical responses in bone around implants: A 3-D nonlinear finite element analysis," *Biomedical Engineering Applications Basis and Communications*, vol. 17, no. 1, pp. 44–49, 2005.

[8] M. Mangalvedhekar and M. Herekar, "The influence of location of oeelusal loading on stresses transferred to implant-supported prostheses and supporting bone: A three dimensional finite element study," *Asian Journal of Midical and Clinical Sciences*, vol. 3, pp. 63–67, 2014.

[9] L. Kong, Y. Zhao, K. Hu, D. Li, H. Zhou, Z. Wu, and B. Liu, "Selection of the implant thread pitch for optimal biomechanical properties: A three-dimensional finite element analysis," *Advances*

in Engineering Software, vol. 40, pp. 474–478, 2009.

[10] L. Kong, Z. Gu, K. Hu, H. Zhou, Y. Liu, and B. Liu, "Optimization of the implant diameter and length in type B/2 bone for improved biomechanical properties: A three-dimensional finite element analysis," *Advances in Engineering Software*, vol. 40, pp. 935–940, 2009.

[11] Y. Sun, L. Kong, K. Hu, C. Xie, H. Zhou, Y. Liu, and B. Liu, "Selection of the implant transgingival height for optimal biomechanical properties: A three-dimensional finite element analysis," *British Journal of Oral and Maxillofacial Surgery*, vol. 47, pp. 393–398, 2009.

[12] J.-R. Xiao, Y.-F. Li, S.-M. Guan, L. Song, L.-X. Xu, and L. Kong, "The biomechanical analysis of simulating implants in function under osteoporotic jawbone by comparing cylindrical, apical tapered, neck tapered, and expandable type implants: A 3-dimensional finite element analysis," *Bitish Journal of Oral and Maxillofacial Surgery*, vol. 69, pp. e273–e281, 2011.

[13] Y. Gao, Y.-F. Li, B. Shao, T. Li, N. Xia, L.-X. Xu, Z.-Y. Wang, and L. Kong, "Biomechanical optimisation of the length ratio of the two endosseous portions in distraction implants: A three-dimensional finite element analysis," *British Journal of Oral and Maxillofacial Surgery*, vol. 50, pp. e86–e92, 2012.

[14] T. Li, X. Yang, D. Zhang, H. Zhou, J. Shao, Y. Ding, and L. Kong, "Analysis of the biomechanical feasibility of a wide implant in moderately atrophic maxillary sinus region with finite element method," *Oral Surgery*, *Oral Medicine*, *Oral Pathology*, *Oral Radiology*, vol. 114, pp. e1–e8, 2012.

[15] M. Mangalvedhekar and M. Herekar, "The influence of location of occlusal loading on stresses transferred to implant-supported prostheses and supporting bone: A three dimensional finite element study," *Asian Journal of Medical and Clinical Sciences*, vol. 3, no. 2, pp. 63–67, 2014.

[16] H. Abuhussein, G. P agin, A. Rebaudi, and H.-L. Wang, "The effect of thread pattern upon implant osseointegration," *Clinical Oral Implants Research*, vol. 21, pp. 129–136, 2010.