International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 07, July - 2017

Compressive Stress, Shear Stress, and Displacement Study on Different Structured Dental Implant: 3-Dimensional Finite Element Analysis

Aswin Yodrux[1] and Nantakrit Yodpijit[2]
[1, 2]Center for Innovation in Human Factors Engineering and Ergonomics, Department of Industrial Engineering,
Faculty of Engineering,
King Mongkut's University Technology North Bangkok,
Bangkok,10800, Thailand

Manutchanok Jongprasithporn[3]
[3]Department of Industrial Engineering,
Faculty of Engineering,
King Mongkut's Institute of Technology Ladkrabang,
Ladkrabang, Bangkok, 10520, Thailand

***Abstract* – This paper presents the use of 3-dimensional finite element analysis (3D-FEA) on dental implant prosthetics. The current research focuses on four patents (three models of United States Patent and Trademark Office (USPTO) and a new conceptual design model) of dental implant threads. The 3D-FEA is performed on dental implant models, with compressive forces of 100 N, and a shear force of 50 N with the force angle of 45 (degree) with the normal line respectively. The compressive stress, shear stress, and displacement analysis is conducted at four different areas, including abutment, implant, cortical bone, and cancellous bone. Findings from this research provide guidelines for new product design of dental implant prosthetics with stress distribution and displacement characteristics. The maximum stress occurred in dental implant prosthetic and the surrounding bone analysis are less than the yield strength of materials. Hence the design is safe.**

Keywords* - *Dental Implant, Stress Distribution , 3-Dimensional Finite Element Method

I. INTRODUCTION

The dental implant is the alternative for patients who have missing teeth or replacing the nature teeth. The implant is made of titanium metals, designed and researched for 30 years. The process made osteocyte adhere as root like Nature and the patients can withstand chewing well. The dental medical care said that the toothache or any oral painful can be applied by 3-dimensional computer aided engineering (3D-CAE) and 3-dimensional finite element methods (3D-FEM) from curing and tooth protecting. Patients need the dental implants for improving their quality of life so, they can chew and taste their food effectively. Most of the users are elderly people. Dental implants must be imported from abroad, of course, the price is very expensive. People who have less income could not effort the dental implants. The initiative our own design in Thailand will reduce cost of the dental implants and increase effort able implants for Thai [1-2].

The propose of this study is to analyse the biomechanical characteristics of the 3-dimensional geometrical model of the dental implant/bone system by using FEM. The compressive stress, shear stress, and displacements analysis are conducted for implant design and original implant design modelling for dental implants [3].

II. METHODS

A. *3-Dimensional Model Design*

The 3-Dimensional geometrical model of dental implant prosthetics system and the surrounding bone (Fig. 1) [4] was created by the 3-dimensional computer aided design (3D-CAD) modelling for testing with 3-Dimensional stress analysis using FEM. CATIA program by Solid Modelling is used to create the 3D-CAD models in this study.

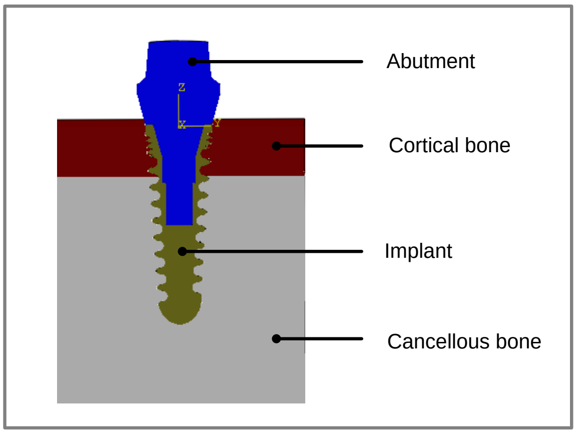

Fig. 1 The geometrical model of dental implant prosthetic systems and the surrounding bone

The current research focused on three patents and one new design (Fig. 2). Three patents of dental implant systems from United States Patent and Trademark Office (USPTO) are including trapezoidal thread-US20140212844A1 [5], reverse buttress thread-US20140816800A1 [6], knuckle thread-US20140147808A1 [7], and new design A thread model.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 07, July - 2017

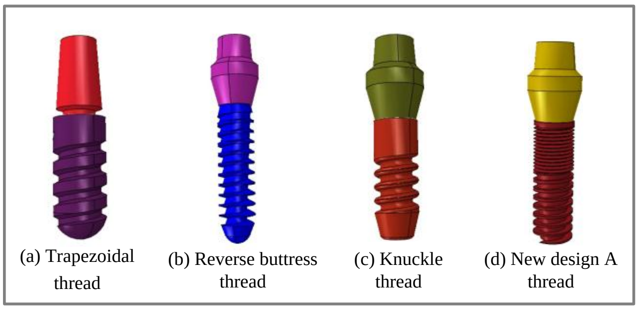

Fig. 2 3-Dimensional model of four dental implant prosthetics

The external thread of four different implants are shown in Fig.3

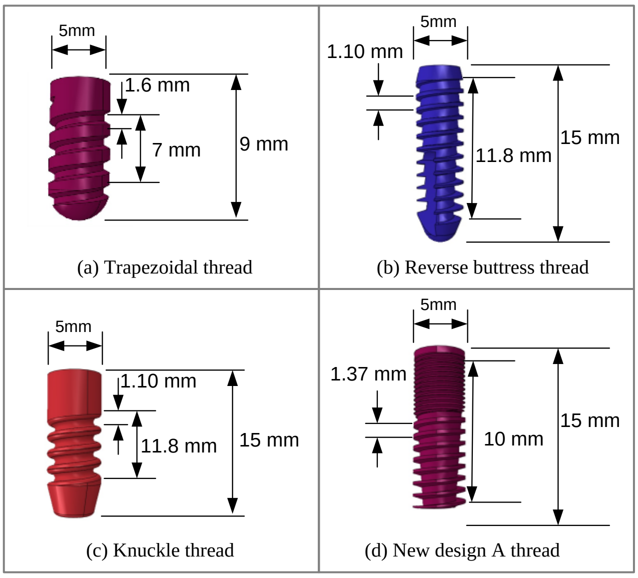

Fig. 3 Thread category of the different implants

B. Materials properties

In the study, Titanium (Ti6Al-4V) was used as implant and abutment materials. All of the materials used in this study were considered to be isotropic, homogeneous, and linearly elastic. The mechanical materials properties were taken from the literature, as shown in Table 1 [8-13].

Table 1 Mechanical properties of materials used in 3D FEM

Materials	Young's modulus, E (MPa)	Poisson's Ratio, ν
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

C. 3-Dimensional Finite Element model

The implant was rigidly anchored in the bone model along its entire interface. The same type of contact was provided at the prosthesis-abutment interface. The 3D-FEA was performed on dental implant models with compressive forces of 100 N, and a sheer force of 50 N with the force angle of 45° with the normal line respectively. The boundary condition was defined to fixed the bone at base along x, y and z of the model, and the adhesion between the implants and bone is shown in Fig. 4.

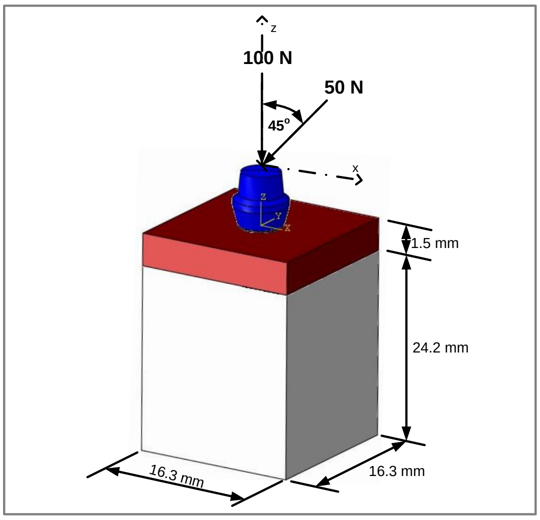

Fig. 4 Cross-sectional view on the symmetry of dental implant with a compressive force and a sheer force

The shape of the 3D-FEA model is shown in Fig. 5

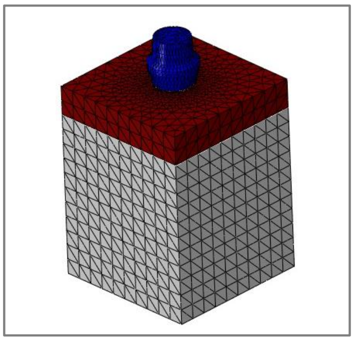

Fig. 5 The 3D-FEM model of dental implant and surrounding bone system

III. Results and Discussions

The stress, shear stress and displacement analysis (using 3D-FEA) were conducted at four different threads areas, including abutment, implant, cortical bone, and cancellous bone.

A. Stress Distribution Analysis

Fig. 6 illustrates the stress distribution in 3D-FEA of four different abutments of dental implant prosthetics under a compressive force.

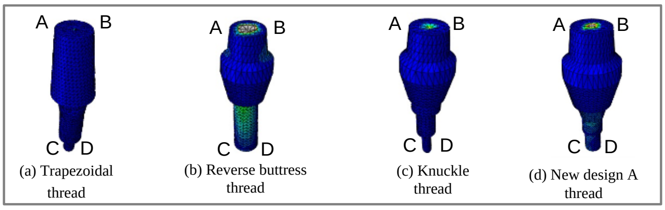

Fig. 6 The stress distribution of four different abutments

Results of the stress in 3D-FEA of four different abutments of dental implant prosthetics under a compressive force are given in Fig. 7. It reveals that the maximum stress of 44.61 MPa is found at point C of trapezoidal thread and the minimum stress of 0.577 MPa is found at point C of new design A thread

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 07, July - 2017

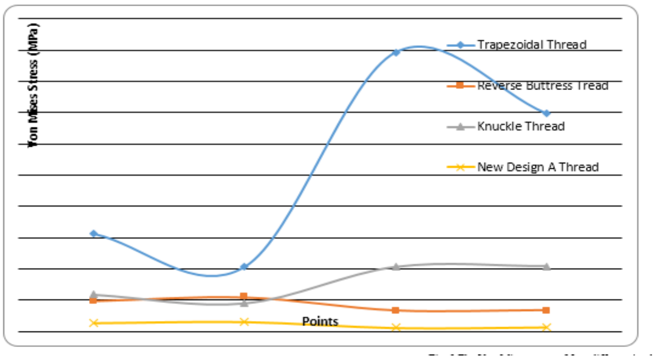

Fig. 7 The Von Mises stress of four different abutments

Fig. 8 illustrates the stress distribution in 3D-FEA of four different implants of dental implant prosthetics under a compressive force.

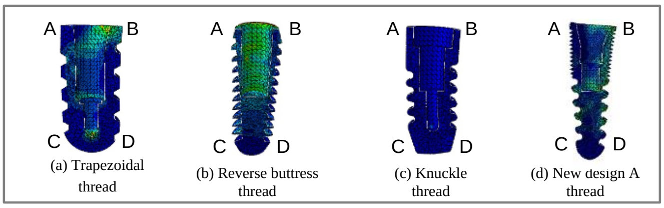

Fig. 8 The stress distribution of four different threads at the implants

Results of the stress in 3D-FEA of four different implants of dental implant prosthetics under a compressive force are given in Fig. 9. It reveals that the maximum stress of 11.73 MPa is found at point B of trapezoidal thread and the minimum stress of 0.019 MPa is found at point C of reverse buttress thread

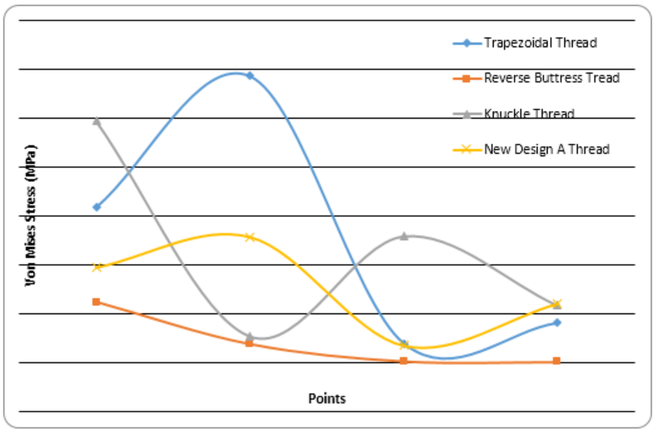

Fig. 9 The Von Mises stress of four different implants

Fig. 10 illustrates the stress distribution in 3D-FEA of four different cortical bone under a compressive force.

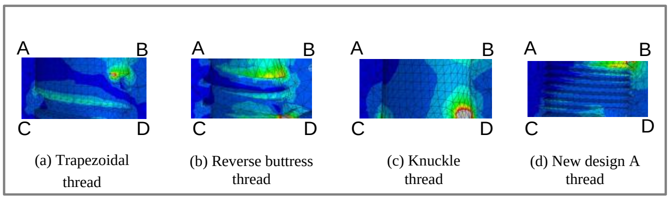

Fig. 10 The stress distribution of four different cortical bone

Results of the stress in 3D-FEA of four different cortical bone of under a compressive force are given in Fig. 11. It reveals that the maximum stress of 10.19 MPa is found at point C of trapezoidal thread and the minimum stress of 0.273 MPa is found at point B of new design A thread

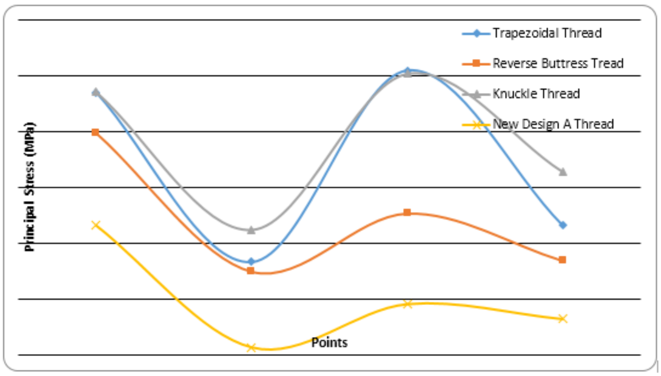

Fig. 11 The principal stress of four different cortical bone

Fig. 12 illustrates the stress distribution in 3D-FEA of four different cancellous bone under a compressive force.

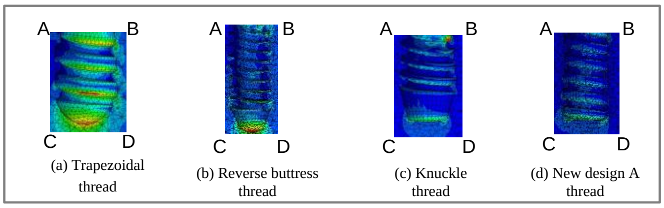

Fig. 12 The principal stress of four different cancellous bone

Results of the stress in 3D-FEA of four different cancellous bone under a compressive force are given in Fig. 13. It reveals that the maximum stress of 15.68 MPa is found at point B of knuckle thread and the minimum stress of 0.0103 MPa is found at point C of reverse buttress thread

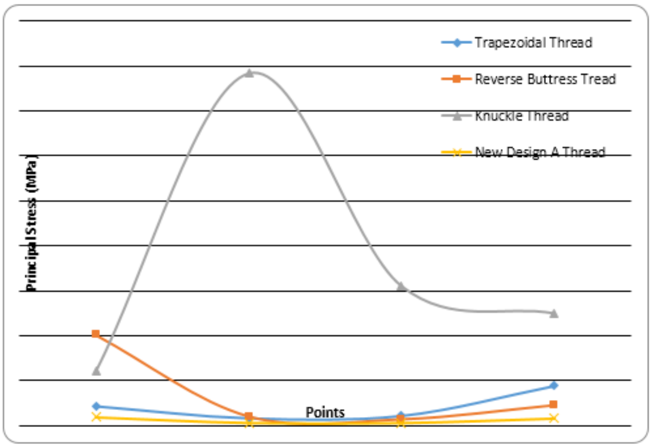

Fig. 13 The principal stress of four different cancellous bone

B. Shear Stress Analysis

Results of shear stress in 3D-FEA of four different abutments under a shear force are given in Fig. 14. It reveals that the maximum shear stress of 44.61 MPa is found at point C of trapezoidal thread and the minimum shear stress of 0.0103 MPa is found at point C of new design A thread

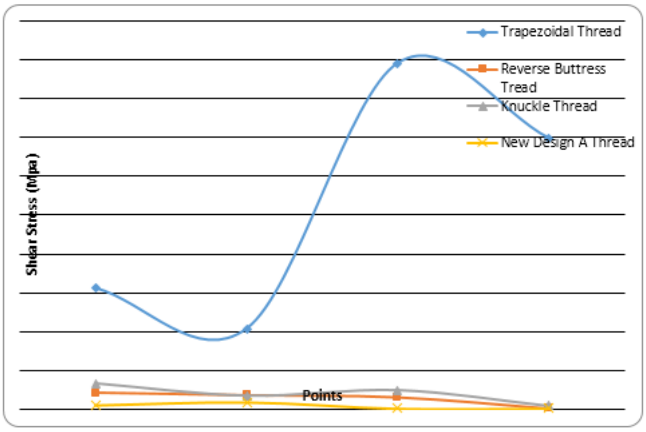

Fig. 14 Shear stress of four different abutments

Results of shear stress in 3D-FEA of four different implants under a shear force are given in Fig. 15. It reveals that the maximum shear stress of 11.73 MPa is found at point B of trapezoidal thread and the minimum shear stress of 0.0086 MPa is found at point B of new design A thread

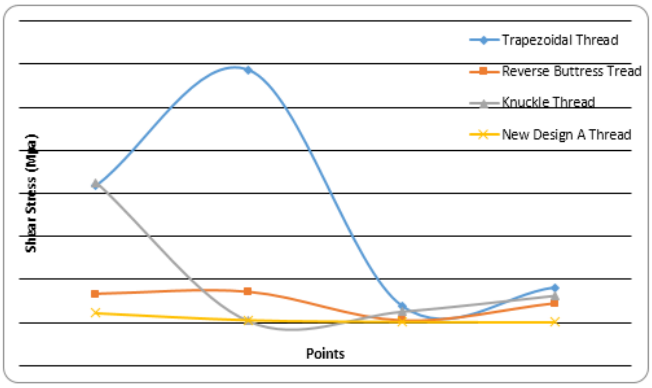

Fig. 15 Shear stress of four different implants

Results of shear stress in 3D-FEA of four different cortical bone under a shear force are given in Fig. 16. It reveals that the maximum shear stress of 10.19 MPa is found at point C of trapezoidal thread and the minimum shear stress of 0.0018 MPa is found at point C of new design A thread

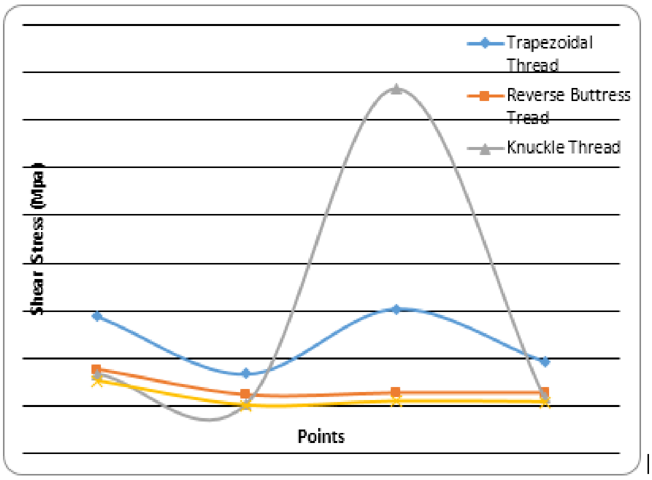

Fig. 16 Shear stress of four different cortical bone

Results of shear stress in 3D-FEA of four different cancellous bone under a shear force are given in Fig. 17. It reveals that the maximum shear stress of 10.28 MPa is found at point B of knuckle thread and the minimum shear stress of 0.0069 MPa is found at point B of new design A thread

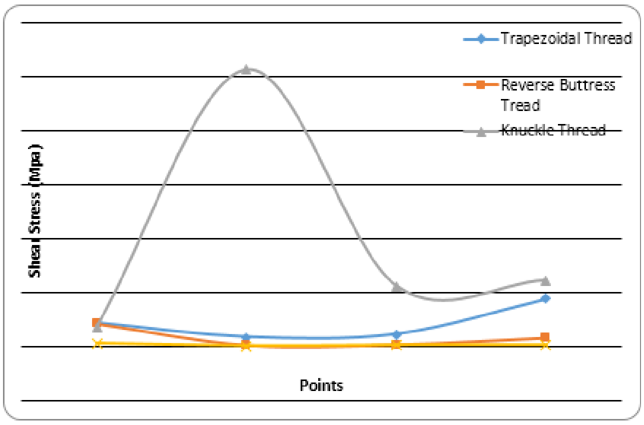

Fig. 17 Shear stress of four different cancellous bone

C. Displacement Analysis

Results of displacement in 3D-FEA of four different abutments under compressive force are given in Fig. 18. It reveals that the maximum displacement of 0.0173 mm is found at point A of trapezoidal thread and the minimum displacement of 0.0155 mm is found at point D of new design A thread

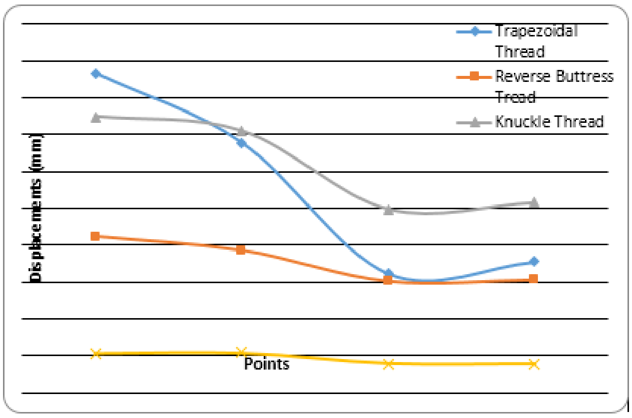

Fig. 18 Displacement of four different abutments

Results of displacement in 3D-FEA of four different implants under compressive force are given in Fig. 19. It reveals that the maximum displacement of 0.0120 mm is found at point A of knuckle thread and the minimum displacement of 0.00118 mm is found at point B of new design A thread

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 07, July - 2017

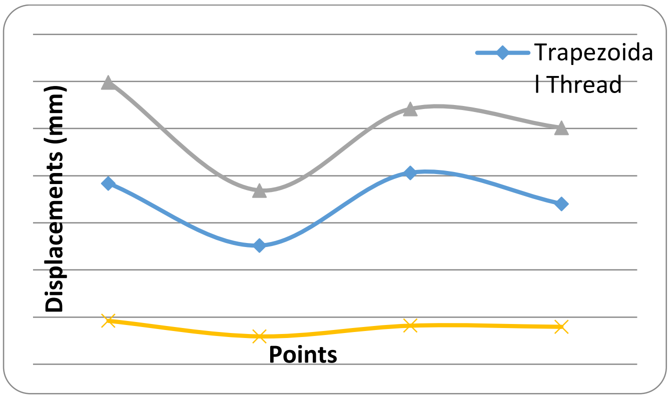

Fig. 19 Displacement of four different implants

Results of displacement in 3D-FEA of four different cortical bone under compressive force are given in Fig. 20. It reveals that the maximum displacement of 0.00926 mm is found at point C of knuckle thread and the minimum displacement of 0.00123 mm is found at point B of new design A thread

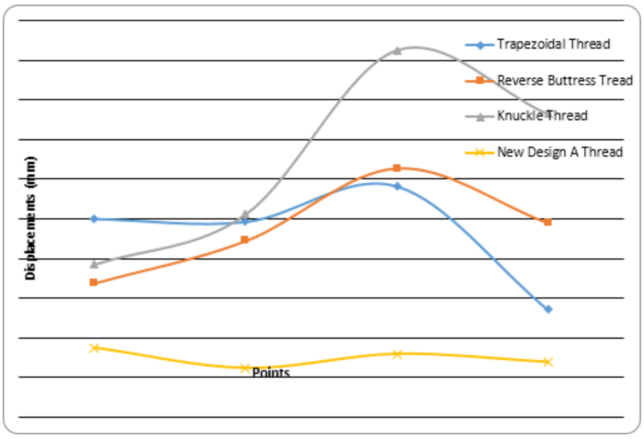

Fig. 20 Displacement of four different cortical bone

Results of displacement in 3D-FEA of four different abutments under compressive force are given in Fig. 21. It reveals that the maximum displacement of 0.0103 mm is found at point D of knuckle thread and the minimum displacement of 0.00135 mm is found at point C of new design A thread

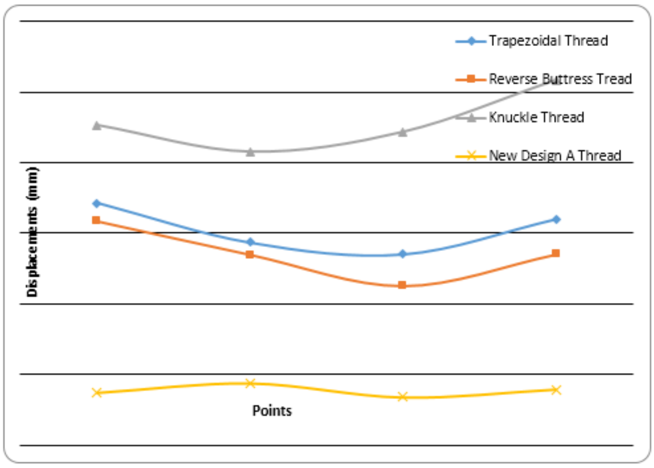

Fig. 21 Displacement of four different cancellous bone

IV. CONCLUSIONS

The 3D-FEA is performed on dental implant models with compressive forces of 100N, and a sheer force of 50 N with the force angle of 45°. In the study, Titanium (Ti6Al-4V) was used as implant and abutment materials. From this research, The maximum stress occurred in four different threads areas, including abutment, implant, cortical bone, and cancellous bone are are less than the yield strength of materials. The study of displacement analysis found that, the minimum displacement of the abutments is found at new design thread, the minimum displacement of the implants is found at new design thread, the minimum displacement of the cortical bone is found at new design thread, and the minimum displacement of the cancellous bone is found at reverse buttress thread. Hence the design is safe.

ACKNOWLEDGEMENTS

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] A.J.C. Trappey, C.V. Trappey, H.-Y. Peng, and T.-M. Wang, "Ontology-based Dental Implant Connection Patent Analysis," Proceedings, The 17th International Conference on Computer Supported Cooperative Work in Design (CSCWD 2013), Whistler, BC, Canada, pp. 257-262, 2013.

[2] S.W.C. Chang, C.V. Trappey, A.J.C. Trappey, and S. Chun-Yi Wu, "Forecasting Dental Implant Technologies Using Patent Analysis," Proceedings of PICMET 14: Infrastructure and Service Integration, 27-31 July 2014, Kanazawa, Japan, pp. 1483-1491, 2014.

[3] I. Roatesi, S. Roatesi, and C. Rotaru, "FEM analysis of one element prosthesis on dental implant," Proceedings, The 5th IEEE International Conference on E-Health and Bioengineering (EHB 2015), Iaúi, Romania, 2015.

[4] G. Zhang, H. Yuan, X. Chen, W. Wang, J. Chen, J. Liang, and P. Zhang, "A Three-Dimensional Finite Element Study on the Biomechanical Simulation of Various Structured Dental Implants and Their Surrounding Bone Tissues", *International Journal of Dentistry., vol. 2016*, pp. 1-9, 2016.

[5] The United States Patent and Trademark Office (USPTO) US20140212844A1.

[6] The United States Patent and Trademark Office (USPTO) US20140816800A1

[7] The United States Patent and Trademark Office (USPTO) US20140147808A1.

[8] 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.

[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.

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 Surg Oral Med Oral Pathol Oral Radiol*, vol. 114, pp. e1-e8, 2012.