11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

NUMERICAL INVESTIGATION OF FATIGUE BEHAVIOR OF DENTAL IMPLANT APPLICATIONS

Hüsna TOPKAYA*[1], Mete Onur KAMAN[2]

[1] Batman University Vocational School Batman, husnatopkaya@gmail.com

[2] Fırat University Mechanical Engineering Department Elazığ, mkaman@firat.edu.tr

Keywords: Dental implant, Fatigue, Finite element method

Dental implant applications are commonly-seen threating method in dentistry [1]. The subjected force to the implant is the main factor that affects the success of threat. The shape of the used-implant affects both the implant stabilization and the bone loss. Implant diameter and length are 3.10 and 13.00 mm respectively. Implant geometry is given below.

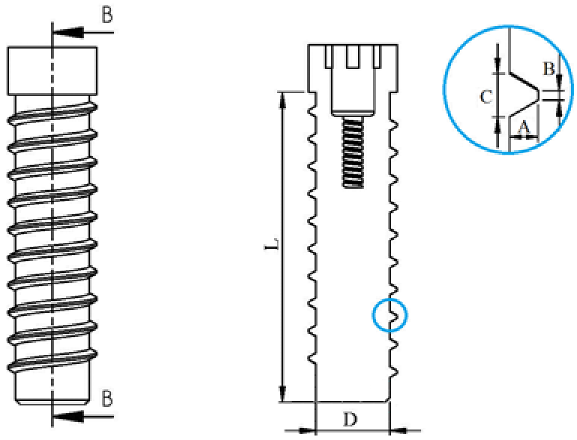

Figure 1 Implant Geometry [2].

In this study, the effect of implant shape on fatigue behavior was investigated by using finite element method. *Ti6Al4V* was used as implant material. Young's Modulus of Ti6Al4V, Cortical Bone and Cancellous Bone are 110 GPa, 13.7 GPa and 1.37 GPa respectively [2,3].

While generating the models, the height of the thread bottom , the height of the thread width and height of the thread varied in the simulations. The models were created by use of SOLIDWORKS base on ISO 14801 [4] criteria and then fatigue analyses were applied to those models by ANSYS software.

The most effective parameter on the life of the implant was found to be the height of the thread, and the least effective parameters were thread bottom height and thread width height. Effect of thread height on fatigue distribution is given in Fig 2.

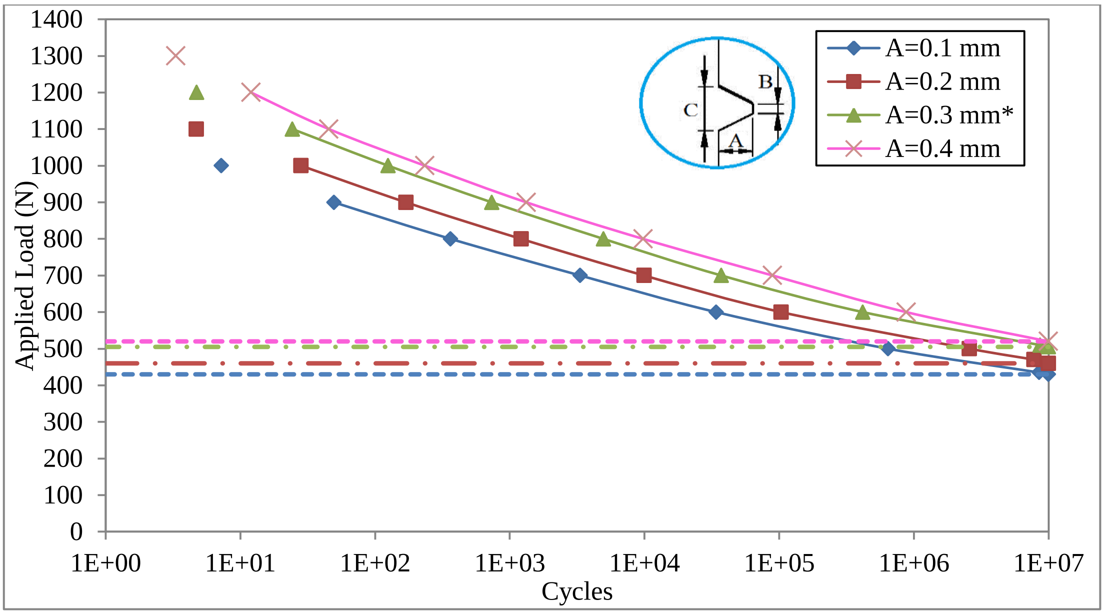

Figure 2 Effect of thread height on fatigue distribution.

It is seen that increase in thread height, thread width and thread bottom height increases the fatigue life of implant. Results showed that fatigue limits of models are 430 N, 460 N, 505 N and 520 N for 0.1, 0.2, 0.3, 0.4 values of A, respectively.

REFERENCES

[1] Bozkaya, D., Muftu, S., Muftu, A., 2004. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis . J Prosthet Dent, 92, 523-30.

[2] Yang j., Xiang Hong-J., 2007. A three-dimensional finite element study on the biomechanical behavior of an FGBM dental implant in surrounding bone, *Journal of Biomechanics*, **40**, 2377–2385.

[3] Lin D., Li Q., Li W., Swain M., 2009. Bone remodeling ınduced by dental ımplants of functionally graded materials, *Journal of Biomedical Materials Research Part B: Applied Biomaterials*, 430-438.

[4] ISO 14801, 2007. International Standard.