World Journal of Medical Sciences 10 (2): 217-221, 2014
ISSN 1817-3055

DOI: 10.5829/idosi.wjms.2014.10.2.82275

Influence of Different Angulations of Force in Stress Distribution in Implant Retained Finger Prosthesis: A Finite Element Study

[1]Pokpong Amornvit, [1]Dinesh Rokaya,
[2]Konrawee Keawcharoen and [3]Nimit Thongpulsawasdi

[1]Maxillofacial Prosthetic Clinic, Department of Prosthodontics,
Faculty of Dentistry, Mahidol University, Bangkok, Thailand
[2]Maxillofacial Surgery Clinic, Golden Jubilee, Medical Centre,
Mahidol University, Nakon Pathom, Salaya, Thailand
[3]Orthopaedic Surgery Clinic, Golden Jubilee, Medical Centre,
Mahidol University, Nakon Pathom, Salaya, Thailand

Abstract: The success in implant retained finger prosthesis is determined by the implant loading and the characteristic of the force is a determining factor in implant loading. The stress distribution varies in finger bone when the loading forces are applied along the various angulations. Aim of this article is to evaluate stress distributions in finger bone when the loading forces are applied along the various angulations of force over the implant using finite element analysis. A finger bone model containing cortical bone and cancellous bone was constructed by using radiograph. Astratech Osssospeed bone level implant of 4.5 mm diameter and 14 mm length was selected for the study. The different angulations of force (0, 30, 60 and 90 degree) were applied to the implant abutment and the stresses generated where analyzed. Results showed when the force was applied at various angulations, the stress generated increased from 0 degree to 90 degree. The maximum stress (124.01 MPa) was at 90 degree force and minimum (31.67 MPa) was at 0 degree force. The maximum stresses were located around the neck of the implant and the cortex bone receives more stress than cancellous bone. So, to achieve long term success, the implant systems must confront biomaterial and biomechanical problems, including in vivo forces on implants, load transmission to the interface and prevent force along the long axis of the implant. Abbreviations: Finite element model (FEM), Finite Element Analysis (FEA), Newton (N), Mega Pascal (MPa), Computer Aided Design (CAD), 3D (3 Dimension).

Key words: Dental Implant・Finger Prosthesis・Stress Distribution・Finite Element Analysis

INTRODUCTION

The finger amputation may be rehabilitated with dental implant-retained finger prosthesis. The biomechanical behavior of implant-retained finger prosthesis plays an important role in its functional longevity inside the bone. Finite Element Analysis (FEA) have been used to study the effects of various shapes of dental implants on distribution of stresses generated in the surrounding bone and to determine an optimal thread shape for better stress distribution. The non-uniformstress pattern at bone and might induce biomechanical overloading failures in implant and bone [1-3].

The characteristic of the force is a determining factor in implant loading. This overloading would cause the micro-damage accumulation at bone and results in bone loss around the neck of the implant [4]. It was reported that the initiated loss of bone mostly around implant neck evolves deeper into the bone [5, 6]. FEA of stress and strain fields have indicated that stress concentration occurring exclusively in the cortical bone near the necks of implants is responsible for the initiation of overload-induced bone resorptionin this region. This analysis technique has been applied to optimize implant design, with an attempt to improve the biomechanical environment in jaw bone/implant systems and reduce bone resorption due to occlusive overload [7-9].

Corresponding Author: Pokpong Amornvit, Maxillofacial Prosthetic Service, Faculty of Dentistry, Mahidol University, 6 Yothi Street, Bangkok 10400, Thailand. Tel: +66-891616260.

Our main purpose is to evaluate stress distributions in the bone and implant under various axis of the loading force, i.e. 0, 30, 60, 60 & 90 degree using finite element analysis in a new three-dimensional model of osseointegrated finger prosthesis.

MATERIALS AND METHODS

Computer Aided Design (Cad) and Finite Elements Modeling: The finger bone model containing cortical bone and cancellous bone was constructed by guiding from radiographs of the metacarpel of finger containing cortical bone and cancellous bone was constructed. A metacarpal block 3696 mm length, 1462 mm width and 1379 mm height was modeled. A Titanium implant model based on bone level implant (Astratech Osssospeed implant system™, Mölndal, Sweden) of 4.5 mm diameter and 14 mm length was selected. The prosthesis system is composed primarily of 3 parts: (a) the implant, (b) the abutment and (c) the abutment screw. The model were designed in SolidWorks 3D software (Solidworks Corporation, Massachusetts, USA) and transported to ANSIS 13 (ANSIS Inc., Southpointe, Canonsburg, PA, USA) as shown in Fig.1 [3].

FEM was created by discrediting the geometric model into smaller and simpler elements. The FEM model consists of total 75713 four-node tetrahedron elements; 3058 elements for cortical bone, 17688 elements for spongy bone, 3169 elements for abutment and 483 elements for the screw and implant. Tetrahedron elements in cortical bone, spongy bone, abutment, screw and implant corresponding to elements in ANSYS element library with each node having three degree of freedom.

The material properties adopted were specified in terms of Young's modulus, Poisson's ratio and density for the implant and all associated components (Table 1) [10]. All materials were assumed to exhibit nonlinear and thermal strain effects.

Loading Conditions: Loading of the implant was done in 3-D with forces of 50 N from top for 1 second in 0, 30, 60 and 90 degrees of the implant simulated pushing action. The end fixed support consisted of the carpel end. The solid model resulting from the intersection of implant and jaw bone represents the assumption of complete osseointegration, restricting any relative displacement between implant and bone. The interface between implant and bone was modeled as a continuous bond. This implies an ideal osseointegration, without any relative motion at the interface. In other words, the implant was rigidly anchored in the bone, showing a fixed and same type of bond at all prosthesis material interfaces.

RESULTS

The obtained results showed the relationship between loads applied on the system, geometrical characteristics of materials and strain. Most frequent used theories for determining stress in bone matrix, Von Mises theory, was applied to this experiment in order to determine stress distribution at the bone-implant interface.The total results of stress and strain were summarized in Fig. 2-3 and Table 2.

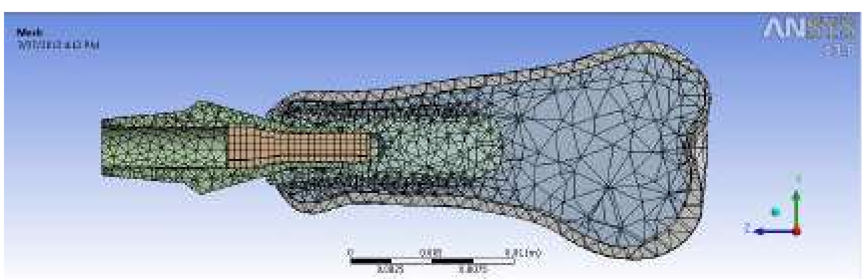

Fig. 1: 3D finite element model of the finger bone containing cortical bone and cancellous bone and the prosthesis system [3].

Table 1: Mechanical properties of materials used in this study [10]

Mechanical Properties of materials used in this study			
Material	Young's modulus (MPa)	Poisson ration (v)	Yield strength (MPa)
Ti-6 Al-4V	110	0.32	800
Cortical bone	14.5	0.323	180
Spongy bone	1.37	0.3	35

Table 2: Stress and Strain distribution under various angulations of force

Degree	Stress (Max) MPa	Areas of maximum stress	Strain (Max) mm/mm	Areas of maximum strain
0	31.673	Implant screw	1.76E-03	Spongy
30	77.221	Abutment	5.16E-03	Spongy
60	113.750	Abutment	7.73E-03	Spongy
90	124.010	Abutment	8.33E-03	Spongy

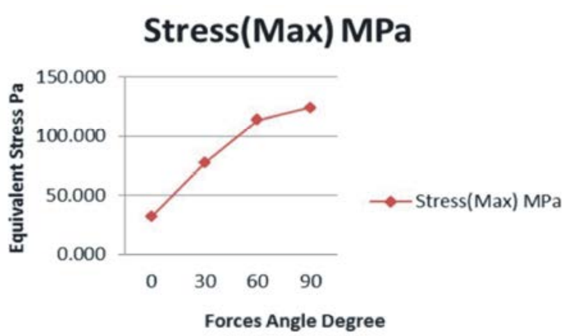

Fig. 2: Values of stress distribution under different angulations of force.

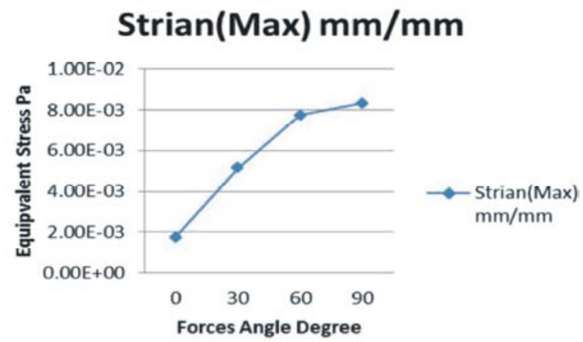

Fig. 3: Values of strain distribution under different angulations of force.

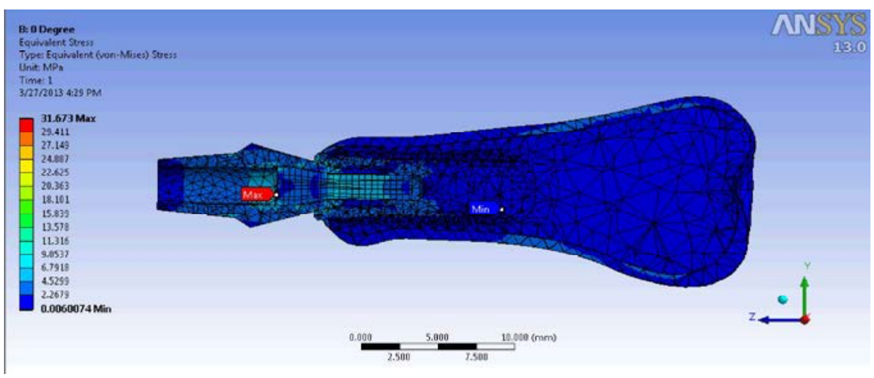

Fig. 4: Stress distribution in the implant and the finger bone loading at 0 degree (Along the long axis of the implant) [3].

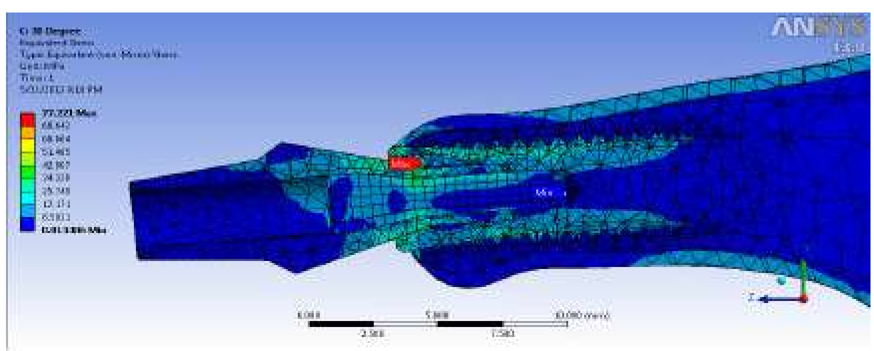

Fig. 5: Stress distribution in the implant and the finger bone loading under 30 degree.

Results showed when the force of 50 N is applied at various angulations, the stress generated increased from 0 degree to 90 degree (Fig. 2 and Table 2). The maximum stress (124.01 MPa) was at 90 degree force and minimum (31.67 MPa) was at 0 degree force. The maximum stresses were located around the neck of the implant (Fig 4-7). The spongy bone receives more stress (Table 2).

When the force was applied from at 0 degree, the stress distribution ranged from 0.0060 to 31.673 MPa (Fig. 4). When the same loading force was applied at

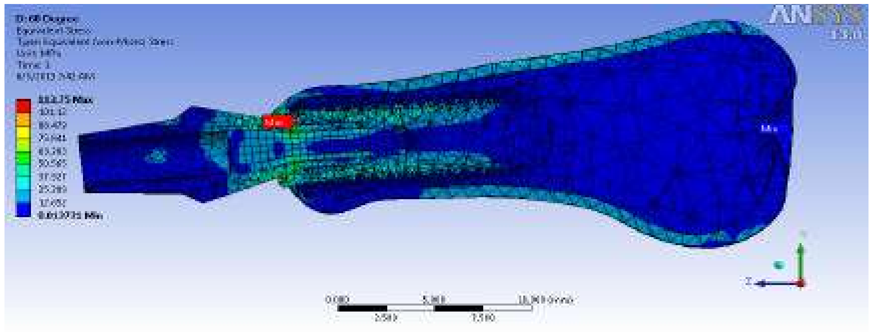

Fig. 6: Stress distribution in the implant and the finger bone loading at 60 degree.

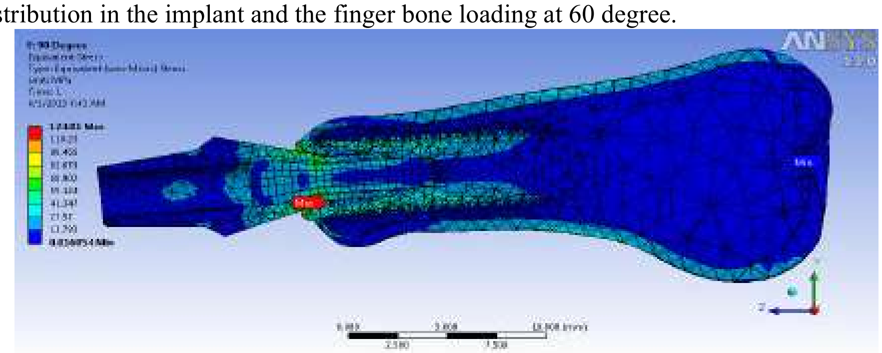

Fig. 7: Stress distribution in the implant and the finger bone loading at 90 degree.

30 degree, the stress distribution ranged from 0.0134 to 77.221 MPa (Fig. 5). When the same loading force was applied at 60 degree, the stress distribution ranged from 0.01372 to 113.75 MPa (Fig. 6). When the same loading force was applied at 90 degree, the stress distribution ranged from 0.016 to 124.01 MPa (Fig. 7).

DISCUSSION

An important factor for the success or failure of a dental implant is the manner in which stresses are transferred to surrounding bone. FEA allows investigators to predict stress distribution in the contact area of implants with cortical bone and around the apex of implants in spongy bone [9, 10]. Regarding the surgical techniques, one-stage technique for the implant placement in implant retained finger prosthesis is safe, reliable and efficient in metacarpal and phalangeal bone if primary stability is optimal [11]. Overload can lead to bone resorption or fatigue failure of the implant, whereas under loading of the bone may causedisuse atrophy and subsequent bone loss. The generation of high stress distribution or concentration inthe bone should be avoided to achieve stable osseointegration for implant restoration. Therefore, overload in a biomechanical systemcauses stress on implant or mechanical componentsleading to bone loss around the implant and/or mechanical failure. The estimation of peri-implant horizontal andvertical bone loss is an important parameter for evaluation andthe implant success [10].

In this study, an implant inserted in a metacarpal for finger prosthesis, has been analyzed using a virtual model. This model is designed in to examine in vitro the effect of the combined dynamicload acting along the long axis on this prosthesis. When the force of 50 N was loading was done from different angulation, the maximum stress (124.01 MPa) was generated when the load was applied at 0 degree (Fig. 4) and the minimum stress (0.006 MPa) was generated when the force was applied at 90 degree (Fig. 7). In a study done by Amornvit *et al*, similar result was noted when the force was applied along the long axis in a finger model [3].When the loading force was applied at 30 and 60 degree, the stress distribution were 77.221 MPa and 113.75 MPa (Fig. 5-6).

Animal experiments and clinical studies have shown that bone loss around implants that may lead to implant failurewas associated in many cases with unfavorable loading conditions [5, 12]. Inappropriate loading causes excessive stress in the bone aroundthe implant and may result in bone resorption. Therefore, it is valuableto investigate the stresses/strains in bone and their relation todifferent parameters of implant and bone.

The CAD model used in this study implied several assumptions regarding the simulated structures. The structures in the model were all assumed to be homogeneous, isotropic and to possess linear elasticity. The properties of the materials modeled in this study, particularly the living tissues, however, are different. Also, it is important to point out that the stress distribution patternsmayhave been different depending on the materials and properties assigned to each layer of the model and the model used in the experiments. Thus, the inherent limitations in this study should be considered.

CONCLUSION

The stress is increased when the force was applied at 0 degree to 90 degree. The maximum stresses were located around the neck of the implant. The cortex bone receives more stress than cancellous bone when the force is given along the long axis of the implant. So, to achieve long term success, the implant systems must confront biomaterial and biomechanical problems, including in vivo forces on implants, load transmission to the interface and prevent force along the long axis of the implant.

REFERENCES

1. Geng J., K.B.C. Tan and G. Liu, 2001. Application of finite element analysis in implantdentistry: a review of the literature. J. Prosthet. Dent., 85: 585-98.
2. Rangert, B., P.H.J. Krogh, B. Langer and N.V. Roekel, 1995. Bending overload and implantfracture: a retrospective clinical analysis. Int. J. Oral Maxillofac. Implants, 10: 326-34.
3. Amornvit, P., D. Rokaya, K. Keawcharoen and N. Thongpulsawasdi, 2013. Stress Distribution in Implant Retained Finger Prosthesis: A Finite Element Study. J. Clin. Diagn. Res., 7(12): 2851-2854.
4. Hansson, S., 2003. A conical implant-abutment interface at the level of the marginalbone improves the distribution of stresses in the supporting bone. Clin. OralImplants Res., 14: 286-93.
5. Hoshaw, S.J., J.B. Brunski and G.V.B. Cochran, 1994. Mechanical loading of Brånemarkimplants affects inerfacial bone modeling and remodeling. Int. J. Oral Maxillofac. Implants, 9: 345-60.
6. Duyck, J., H.J. Ronold, O.H. Van, I. Naert, S.J. Vander and J.E. Ellingsen, 2001.The influence of static and dynamic loading on marginal bone reactions aroundosseointegrated implants: an animal experimental study. Clin. Oral Implants Res., 12: 207-18.
7. Pierrisnard, L., G. Hure, M. Barquins and D. Chappard, 2002. Two dental implants designedfor immediate loading: a finite element analysis. Int. J. Oral Maxillofac. Implants, 17: 353-62.
8. Qian, L., M. Todo, Y. Matsushita and K. Koyano, 2009. Effects of implant diameter, insertiondepth and loading angle on stress/strain fields in implant/jawbone systems. Int. J. Oral Maxillofac. Implants, 24: 877-86.
9. Meijer, H.J.A., J.H. Huiper, F.J.M. Starmans and F. Bosman, 1992. Stress distribution arounddental implants: influence of superstructure, length of implants and height of mandible. J. Prosthet. Dent., 68: 96-102.
10. Roos, J., L. Sennerby and T. Albrektsson, 1997. An update on the clinical documentation on currently used bone anchored endosseous oral implants. Dent. Update, 24(5): 194-200.
11. Amornvit, P., D. Rokaya, K. Keawcharoen, S. Raucharernporn and N. Thongpulsawasdi, 2013. One-Vs two-stage Technique for Implant Placement in Finger Prosthesis. J. Clin. Diagn. Res., 7(9): 1956-1958.
12. Natali, A.N., P.G. Pavan and A.L. Ruggero, 2006. Evaluation of stress induced in peri-implant bone tissue by misfit in multi-implant prosthesis. Dent. Mater., 22: 388-95.
13. Quirynen, M., I. Naert and D. van Steenberghe, 1992. Fixture design and overload influence marginal bone loss and fixture success in the Brånemark System. Clin. Oral Implants Res., 3: 104-11.