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

Pokpong Amornvit, Dinesh Rokaya, Konrawee Keawcharoen and Nimit Thongpulsawasdi

Maxillofacial Prosthetic Clinic, Department of Prosthodontics, Faculty of Dentistry, Mahidol University, Bangkok, Thailand
Maxillofacial Surgery Clinic, Golden Jubilee, Medical Centre, Mahidol University, Nakon Pathom, Salaya, Thailand
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 Ossospeed 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-uniform stress 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 resorption in 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].
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 Ossospeed 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 discretizing 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.

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

<table>
<thead>
<tr>
<th>Material</th>
<th>Young's modulus (MPa)</th>
<th>Poisson ration (v)</th>
<th>Yield strength (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ti-6 Al-4V</td>
<td>110</td>
<td>0.32</td>
<td>800</td>
</tr>
<tr>
<td>Cortical bone</td>
<td>14.5</td>
<td>0.323</td>
<td>180</td>
</tr>
<tr>
<td>Spongy bone</td>
<td>1.37</td>
<td>0.3</td>
<td>35</td>
</tr>
</tbody>
</table>

Fig. 1: 3D finite element model of the finger bone containing cortical bone and cancellous bone and the prosthesis system [3].
Table 2: Stress and Strain distribution under various angulations of force

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

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
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 and the 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 dynamic load 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 failure was associated in many cases with unfavorable loading conditions [5, 12]. Inappropriate loading causes excessive stress in the bone around the implant and may result in bone resorption. Therefore, it is valuable to investigate the stresses/strains in bone and their relation to different 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 patterns may have 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**