Comparative Evaluation of Stress Distribution within Computerized Bone Model using Four Different Implant Collar Designs: A Three-dimensional Finite Element Analysis

1Amit B Jadhav, 2Pronob K Sanyal, 3Siddharth S Gosavi, 4Abhijeet R Kore, 5Swapnil S Jadhav

ABSTRACT

Aim: To evaluate the effect of four types of implant collar designs on stress distribution in a computerized bone model under vertical and angular load and also to evaluate which platform design evokes a better response on a mechanical basis.

Materials and methods: A finite element model of threaded implant with four different kinds of platform designs (divergent, straight, convergent, and step) with their corresponding superstructure embedded within the bone was created. Different test conditions incorporating the four types of platform designs under separate 500 N, axial, and 45° oblique forces were created to investigate the stress distribution within the computerized bone model. The three-dimensional finite analysis study was selected since it is useful in determining stress distribution around the dental implant and also bone response to vertical and angulated load.

Results: Divergent collar design shows minimum and convergent collar design shows maximum stresses concentrated at cortical bone for all collar designs under vertical and oblique loading.

Conclusion: Divergent collar design resists the crestal bone loss, gives better response on a mechanical basis, and may be more suitable in both D2 and D3 types of bones.

Keywords: Implant collar design, Stress distribution, Three-dimensional finite element analysis.


Source of support: Nil

Conflict of interest: None

INTRODUCTION

The ideal goal of modern dentistry is to restore the patient to normal function, comfort, esthetics, speech, and health.1 Though the science of restoration of missing teeth is as old as 300 BC with the Egyptians employing a variety of methods to secure the prosthetic teeth, the successful replacement of lost natural teeth by dental implant is a major advance in dentistry.2

The success or failure of dental implants depends on various complex interwoven factors such as bone quantity, bone quality, surgical techniques, implant designs, and surface- and host-related factors, etc.3 As a result of continued research, diagnostic tools, treatment planning, implant designs, materials, and techniques, predictable success is now a reality for the rehabilitation of many challenging clinical situations.1 Long-term success rates as high as 95% for mandibular implants and 90% for maxillary implants have been reported.3

Finite element model has become one of the most successful engineering computational methods and most useful analysis tool since the 1960s.4,5 It is showing overwhelming capability and versatility in its application in dentistry.6,7 The basic concept is that a body or a structure may be divided into smaller elements (“finite elements”) connected at a finite number of joints called “nodes” (“nodal points”). Calculations are formulated and combined to obtain the solution for the entire body or structure.

So the purpose of the study was to evaluate the stress distribution within two different bone densities (D2, D3) by using four different implant collar designs (straight, divergent, convergent, and step) to evaluate which collar design causes minimum crestal bone loss, in order to give long-lasting restoration to the patient.

AIMS AND OBJECTIVES

• The aim of this study was to evaluate the effect of four types of collar design on stress distribution in a computerized bone model under vertical and angular load and also to evaluate which platform design evokes a better response on a mechanical basis.
• Evaluation of stress distribution of divergent, straight, convergent, and step collar designs in a computerized bone model under vertical and angular load.
• Comparison of stress distribution of divergent, straight, convergent, and step collar designs in a computerized bone model under vertical and angular load.

MATERIALS AND METHODS

This study was conducted using a three-dimensional (3D) finite element model to evaluate the pattern of stress distribution in a mandibular section of bone with a missing first molar restored with an implant-supported all-ceramic crown.

The study was carried out under the following steps:
• Construction of the geometric model.
• Preparation of the finite element mesh.
• Application of the material properties.
• Application of boundary conditions.
• Application of different loads.
• Analysis of stress pattern.

Construction of the Geometric Model

Modeling of the Bone

D2 and D3 bones with similar anatomy as that of the human mandible representing the section of the mandible in the first molar region were modeled with cortical and cancellous bone. The only difference between the D2 and D3 model was thickness of the cortical bone, which is 2 and 1 mm respectively.

Modeling of Implant Body

A straight implant with the dimensions 4×15 mm was used. The implant was divided into implant body and implant collar, with lengths 13 and 2 mm respectively. The implant body had a coronal diameter of 4 mm and apical diameter of 3.5 mm.

All implant models had buttress thread design of spiral type along the implant body length with following specifications:
• Thread pitch: 0.8 mm
• Thread depth: 0.36 mm
• Thread width: 0.25 mm
• Apical face angle: 45°
• Thread helix angle: 7°.

Modeling of Implant Collar

The four different implant collar designs modeled were straight, divergent, convergent, and the new step design (Fig. 1). Each implant collar had the same diameter at the implant collar interface (4 mm) and same height (2 mm).

Only the diameter at the collar abutment interface was different, as follows:
• Straight: 4 mm
• Convergent: 3.5 mm
• Divergent: 4.5 mm
• Step: 4.5 mm

All implant collar designs had microgrooves of parallel type along their length. Implant collar features a 0.2 mm machined coronal aspect followed by a 1.8 mm surface with microgrooves. The six microgrooves are circumferential with a depth of 0.06 mm, pitch of 0.3 mm, and width of 0.02 mm.

Modeling of Implant Abutment

A cone-shaped convergent implant abutment was modeled having a diameter of 3 mm at the abutment collar interface and a height of 5 mm.

Modeling of Crown

Full ceramic crown for mandibular 1st molar was modeled. After modeling all the models were assembled to form a single unit.

Preparing of the Finite Element Mesh

All assembled units were subjected to meshing with the help of hypermesh 11 software. The models showed a number of elements ranging from 500,718 to 550,898 and a number of nodes ranging from 92,228 to 100,312.

Application of the Material Properties

All materials used in the model, such as vital tissues (cortical, cancellous bone) and implant with superstructure were presumed to be homogenous, isotropic, and linear elastic. The Poisson’s ratio (v) and Young’s modulus (E) of elasticity of the material were incorporated into the model.
The corresponding elastic properties, such as Young’s modulus and Poisson’s ratio were determined according to literature survey.³

**Application of Boundary Conditions**

The model was fixed at the base. A support was provided at the inferior surface of the model and also at the distal ends of the mandibular section to simulate the action of the muscles and ligaments.

**Application of Different Loads**

The applied forces were static. Vertical as well as oblique forces were considered as the latter represent more realistic occlusal forces.³ Loads applied were as follows:

- Vertical forces of 500 N was applied at centric fossa.⁸
- Oblique forces of 500 N was applied along the lingual inclination of buccal cusps in the linguobuccal direction at 30° angle.⁹

**Analysis of Stress Pattern**

The Von Mises stress analysis was done. A total of four models of implant were prepared, which were tested for Von Mises stress by applying vertical and angular load of 500 N in D2 and D3 types of bone. Von Mises stress values are defined as the beginning of the deformation for ductile materials, such as metallic implants. Failure occurs when Von Mises stress values exceed the yield strength of an implant material. Therefore, they are important for interpreting the stresses occurring within the implant material.

**RESULTS**

The stress analysis executed by Ansys software provided results that enabled the tracing of Von Mises stress field in the form of color-coded bands. Each color band represents a particular range of stress value, which is given in mega-Pascals (MPa). Blue and red colors represent minimum and maximum stress respectively. It was found that stresses were concentrated at crestal bone and implant collar for all models.

Figures 2A to D shows Von Mises stress generated within the bone for a straight collar design in D2 quality bone during vertical and oblique loading. In D2 type of bone, maximum stresses were 23.91 and 135.45 MPa observed at cortical bone and 45.31 and 494.50 MPa.
observed at implant collar under vertical and oblique loading respectively.

Table 1 shows Von Mises stress at implant collar in D2 and D3 bone during vertical and oblique loading. Results with green color show minimum stress and with red show maximum stress. Results do not show any significant difference between the stresses obtained for each collar design.

Table 2 shows Von Mises stress at crestal bone in D2 and D3 bone during vertical and oblique loading. Divergent collar shows minimum stress and convergent collar shows maximum stress. Results show a significant difference for each collar design. Divergent collar design shows minimum and convergent collar design shows maximum stresses concentrated at the cortical bone for all collar designs under vertical and oblique loading.

DISCUSSION

Micromovement of an endosteal dental implant and excessive stress at the implant bone interface have been suggested as potential causes for peri-implant bone loss and failure of osseointegration. In a 3-year longitudinal study of successful dental implants, Van Steenberghe et al reported an average loss of marginal bone of 0.4 mm during the first year following implant placement and 0.03 mm per year during the second and third years.

In this study all the optimum conditions were taken into consideration. First of all bone conditions D2 and D3 were chosen as these types of bone most commonly are found in the mandibular posterior region. In D2, a thick layer (2 mm) of compact bone surrounds a core of dense trabecular bone. In D3, a thin layer (1 mm) of cortical bone surrounds a core of dense trabecular bone of favorable strength.

Commercially pure titanium is used as the implant material as it is the most biocompatible material used for any endosseous implant. Kong et al considered 0.8 mm as the optimal thread pitch for achieving primary stability and optimum stress production and the optimal thread height ranged from 0.34 to 0.5 mm and thread width between 0.18 and 0.3 mm, with thread height being more sensitive to peak stresses than thread widths. Therefore, a pitch of 0.8 mm, thread depth of 0.36 mm, and width of 0.25 were selected for the implant thread design.

All four types of implant collar designs had parallel-type microgrooves along their length. Previous studies have shown that the application of microgrooves to the implant surface can direct cellular morphology and cell migration and improve cell adhesion.

The finite element model created in this study was a multilayered complex structure involving a solid implant and a layered specific crown. It is important to note that the stress in different bone qualities may be influenced greatly by the materials and properties assigned to each layer.

The results of this study showed that the maximum Von Mises stresses were observed at the crestal region or the neck of the implant for both threaded and cylindrical types in all the four densities of bone. This is similar to the results obtained by various other studies that demonstrated that bone loss begins around the implant neck due to higher bone stresses at the crestal region.

It was observed that the stresses generated at the cortical bone were more in the convergent collar design and minimum in the divergent collar design under vertical and oblique loading in D2 and D3 bone quality (Graphs 1A and B). This is in similar to the results obtained by Shen et al in 2010.

LIMITATIONS

- The structures in the model were all assumed to be homogenous and isotropic and to possess linear elasticity → transversely isotropic and nonhomogeneous.
- Cement thickness layer was not taken into consideration.

CONCLUSION

Within the limitations of this study, the following conclusions were drawn:

- Maximum Von Mises stress was observed at the crestal region of the bone and collar region of the implants in all the models.
- Divergent collar design results in minimum crestal bone loss during vertical loading and oblique loading at cortical bone and during vertical loading at implant collar in D2 and D3 types of bone quality.

<table>
<thead>
<tr>
<th>Table 1: Von Mises stress at implant collar</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collar design</td>
</tr>
<tr>
<td>Straight</td>
</tr>
<tr>
<td>Divergent</td>
</tr>
<tr>
<td>Convergent</td>
</tr>
<tr>
<td>Step</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2: Von Mises stress at cortical bone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Collar design</td>
</tr>
<tr>
<td>Straight</td>
</tr>
<tr>
<td>Divergent</td>
</tr>
<tr>
<td>Convergent</td>
</tr>
<tr>
<td>Step</td>
</tr>
</tbody>
</table>
Convergent collar design results in maximum crestal bone loss in all the models during both vertical and oblique loading in D2 and D3 quality bone except for oblique loading at implant collar.

In view of the above conclusions it may be inferred that the divergent collar design resists the crestal bone loss, gives better response on a mechanical basis, and may be more suitable in both D2 and D3 types of bone.

REFERENCES


