Finite Element Simulation of Dental Implantation Process

Rudi van Staden ¹, Hong Guan ¹, Yew-Chaye Loo ¹, Newell Johnson ² and Neil Meredith ³

¹Griffith School of Engineering, Griffith University Gold Coast Campus, Australia
²School of Dentistry and Oral Health, Griffith University Gold Coast Campus, Australia
³Neoss Pty Ltd, Harrogate, United Kingdom

Email contact: Y.Loo@griffith.edu.au

ABSTRACT

Using the finite element procedure, the insertion process of a dental implant into a section of the mandible is analysed. Elevated stress concentrations in both cancellous and cortical bone are examined during implantation. An implant of length 11mm is inserted at 1mm increments into the mandible. The torque applied to the implant top ranges from 0 to 450Nmm for a total time period of 36 seconds. The complex geometrical and material properties of the implant and mandible are modelled using three dimensional (3D) brick elements. Nonlinear material properties are assigned for both cancellous and cortical bone. The assumptions made in the analysis include: (a) simulating and replicating the implantation process as a step-wise instead of a continuous process; (b) parallel threaded implant is used and the implant does not rotate during insertion into the mandible. The optimum insertion torque and the set time periods for each level of torque applied are found to be the controlling factors that would help produce ideal stresses within the jawbone throughout the entire implantation process.

Key words: Dental implant, Implantation procedure, Finite element technique

1 Introduction

Modern dental implant is a biocompatible titanium device surgically placed into a jawbone to support a prosthetic tooth crown in order to replace missing teeth. Implants are superior to conventional prostheses, in both function and long-term predictability. The process of implantation requires an optimum stress profile in order to maintain a strong and healthy jawbone: a stress that is too high may cause irreversible damage to the jawbone; one that is too low may fail to stimulate the jawbone sufficiently for satisfactory healing of the wound and thus, for osseointegration. Implants should ideally function for life. Without the correct implantation technique an implant may fail shortly after insertion. The method by which the implant is placed into the jawbone plays a vital role for it to be biologically accepted by the surrounding jawbone. Clinical observations have indicated that there is a direct relationship between the insertion torque, the insertion speed and the success rate of implantation [1].

The Finite Element Method (FEM) has been used extensively in implant research [2,3], and has proven to be reliable in analysing the stress distributions in surrounding tissues. Extensive research has been conducted into implant design, percentage of osseointegration, various loading scenarios and implant orientation [4-10]. Research into the simulation of the dynamic implantation process is non-existent. It
is essential for the dentists to have a thorough understanding of the stress characteristics within the jawbone to guarantee a successful implantation. In this study Strand7 [11] Finite Element Analysis (FEA) system is used to model and analyse the implantation process in a step-wise manner.

A successful implantation is governed by various factors. These include the torque applied to the implant, the insertion speed, the preparation of the site prior to insertion and the clinical health of the patient. A study of 23 patients by Ottoni et al. [1] found that among 10 failed implants, 9 were placed with an insertion torque of 200Nmm. This study concluded that the survival rate was independent of implant length, site position, and bone quality and quantity. A further conclusion indicated that to achieve osseointegration the insertion torque must exceed 320Nmm. Schmid et al. [12] found that 80% of the highest torque was applied during the final few turns of implantation. They also found that 15-20% of the highest torque was applied at the initial stages of implantation. The ultimate aim of implantation is to ensure that the bone in particular cancellous does not exhibit traumatic fracturing. This implies that an ideal stress level within the cancellous bone should be maintained during and after implantation to ensure optimum wound healing. Published literature [7, 8] indicated that a stress level between 250psi (1.72MPa) and 400psi (2.76MPa) within the surrounding cancellous bone is ideal for bone growth. This stress level is used as a reference to evaluate the results obtained in this study.

2 The Finite Element Model
This paper aims to replicate the implantation process using a simplified yet efficient modelling approach. This objective is achieved by firstly constructing a 3D model of the implant itself, followed by those for the cancellous and cortical bone. Figure 1 (a) shows the implant model consisting of 2404 brick (8-node hexahedral and 6-node wedge) elements and 3912 nodes. The implant is cylindrical without taperage and is parallel threaded with 1mm pitch thread. Figure 1 (b) shows the dimensions for the jawbone section based on Computer Tomography (CT) scanned images of a posterior section of the mandible. The cancellous bone has 14078 brick elements and 25312 nodes. The cortical bone has a set thickness of 1.3mm [13] with 1940 brick elements and 4280 nodes. The total numbers of elements and nodes for the entire implant and mandible model when the implant is fully inserted are 18422 and 33504 respectively. The present study has increased complexity in its finite element model in that the total numbers of elements and nodes are more than doubled than those used in previous studies, for example, by Lewinstein et al. [14].

(a) Dimensions of implant (b) Dimensions of jawbone

Figure 1 Finite element model of implant and jawbone
An important criterion for simulating a realistic implantation process is the loading and boundary conditions – i.e. the applied torque and restraints. Figures 2 (a), (b) and (c) show the time dependent torque that is positioned at the top of the implant and the fixed constraints in the distal direction. The torque is replicated by applying four concentrated tangential forces, as shown in Figure 2 (b). Details of the time dependant torque are discussed further in Section 3.

![Diagram showing loading and boundary conditions](https://example.com/diagram.png)

(a) Torque applied to implant model  (b) Equivalent tangential forces applied to implant top  (c) Location of fixed constraints

Figure 2 Load and boundary conditions applied to the implant and jawbone

The material properties of implant and jawbone used in this study are shown in Table 1. The implant properties are determined based on the commonly used values for titanium, as indicated in previous studies [4-6, 9, 10, 15-17]. The Young’s modulus of the jawbone as documented in previous literature [4-6, 9, 10, 15-20] ranges from 7.5MPa to 20MPa. In this study, the average values of Young’s moduli of cancellous and cortical bone are taken. Table 1 also gives the assumed properties of blood interface.

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Implant</th>
<th>Cancellous Bone</th>
<th>Cortical Bone</th>
<th>Blood Interface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young Modulus, E (GPa)</td>
<td>102</td>
<td>9.5</td>
<td>13</td>
<td>0.07</td>
</tr>
<tr>
<td>Poissons Ratio, ν</td>
<td>0.3</td>
<td>0.3</td>
<td>0.35</td>
<td>0.3</td>
</tr>
<tr>
<td>Density, $\rho$ (x10^6 kg/mm³)</td>
<td>4.54</td>
<td>1.5</td>
<td>1.5</td>
<td>1.5</td>
</tr>
</tbody>
</table>

Unlike previous FEA, non-linear material properties are assumed in this study for both cancellous and cortical bone. It is expected that both would exceed their respective maximum yield stress during implantation. Hence the stress-strain relationships are defined up to the point of fracture (see Figures 3 (a) and (b) for cancellous and cortical bone, respectively).

The relationship between the amount of fractured bone and the percentage of osseointegration is still unknown. For the purpose of this study it is assumed that the thickness of the blood and bone fragments is 0.5mm filling around the implant threads (Figure 4). The dimensions and location of the fragment interface plays an important role in the stress characteristics within both cancellous
and cortical bone. It is assumed that, at each stage of insertion, the bottom two threads of the implant cut the bone so that the remaining upper threads come in contact with the blood and bone interface. During implant insertion, the von Mises stresses at specific brick elements, B_{CAN} and B_{COR}, (Figure 4) are recorded under increased torque to offer a better understanding of the stresses experienced by specific regions of the jawbone where a stress concentration occurs [21].

3 Step-wise Analysis

Ideally the insertion process should be modelled as a continuous one. However modelling difficulties such as auto re-meshing would have been encountered if a continuous process were to be modelled. Hence a simplified modelling approach is proposed in this study. A series of finite element models are constructed and simulated to replicate the implantation process in a step-wise manner. In other words, an individual model is constructed for each new insertion depth of the implant. Each model differs from the preceding one by the fact that the implant is inserted 1mm deeper into the jawbone. As the implant is 11mm long, there are eleven different models to be analysed. When the implantation process is initiated the implant is assumed to be inserted 0.5mm into the cortical bone thus replicating the implant tip being pushed slightly into the top surface of the bone prior to the application of any torque. As the depth of the implant into the jawbone increases, the area of blood and bone interface also increases which drastically alters the von Mises stresses within the jawbone.

![Figure 3](image1.png)

**Figure 3** Stress-strain relationships [20]

![Figure 4](image2.png)

**Figure 4** Location of fragments of blood and bone

![Figure 5](image3.png)

**Figure 5** Torque vs. time during implantation
For each of the eleven models, the time periods for each level of the applied torque must be defined. This is shown in Figure 5. Setting the correct torque for each time period is difficult because during manual implantation the torque at one specific time is distinct for each implant specialist. The specialist has his/her own opinion and judgement in determining the level of torque for different bone qualities and quantities. The torque over a set period of time as presented in Figure 5 is based on the recommendations given by a number of implant specialists. Due to the fact that a time dependent torque is applied to each one of the eleven models, a non-linear transient dynamic analysis must be performed. In these analyses, the material nonlinearity is also taken into account according to the stress-strain relationships shown in Figure 3.

Figure 5 indicates that the torque applied increases rapidly from 0 to 450Nmm during the first 8 seconds (or 2mm implant insertion into the jawbone). As the implant is being screwed into the jawbone, the surface area contact between the implant and surrounding bone increases leading to a higher degree of resistance to the implant entering the jawbone. Thus a larger level of torque is required to insert the implant at a constant speed. The torque then remains constant at 450Nmm up to 23 seconds (or 7mm implant insertion) because the resistance to the torque does not vary significantly. During the final 12 seconds of implantation, the torque increases up to a maximum of 1000Nmm. Findings by Schmid et al. [12] confirm the torque vs. time characteristics in that the highest applied torque is used during the final few turns of the implant. The assumed torque vs. time relationship is also supported by Ottoni et al. [1] who stated that in order to achieve osseointegration the level of applied torque must exceed 320Nmm. Hence the torque used in this analysis allows for the optimum osseointegration to occur.

4 Stress Variation at Specific Jawbone Locations
The von Mises stresses recorded at locations B_{CAN} and B_{COR} for all insertion stages are plotted against the insertion time (0-36 seconds) (see Figures 6 and 7). Note that the insertion depths are also indicated in the figures. The corresponding stress characteristics at B_{CAN} are described in detail in Table 2.
Table 2  Explanation of stress characteristics at $B_{CAN}$

<table>
<thead>
<tr>
<th>Insertion depth (mm)</th>
<th>Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>The stress magnitude is relatively low because there is no direct contact between the implant and cancellous bone. The stress increases as the applied torque increases.</td>
</tr>
<tr>
<td>1</td>
<td>The bottom thread comes into direct contact with the cancellous bone leading to a sudden increase in the stress.</td>
</tr>
<tr>
<td>2</td>
<td>The stress increases markedly and more rapidly at this stage because the cancellous bone is in contact with the bottom two threads without the blood and bone interface. However the cortical bone comes into contact with the interface hence increasing the stress within the cancellous bone.</td>
</tr>
<tr>
<td>3 - 7</td>
<td>The torque is constant which leads to no change in the stress level for each insertion depth. More surface area contact between the implant and cancellous bone leads to a gradual decrease in the stress level for each stage of insertion.</td>
</tr>
<tr>
<td>8 - 9</td>
<td>The combination of an increased torque and surface area contact between the implant and cancellous bone are responsible for the increase in stress levels.</td>
</tr>
<tr>
<td>10</td>
<td>Initially at 669Nmm torque a decrease in stress occurs due to the fact that the cortical bone comes into contact with the implant neck thus increasing the stress in the cortical bone while reducing that in the cancellous bone.</td>
</tr>
<tr>
<td>11</td>
<td>Finally a further decrease in stress is observed at this stage because the cortical bone absorbs a higher stress. A decrease in stress from 9mm to 11mm of insertion is due to the fact that more implant surface area is in direct contact with cancellous bone.</td>
</tr>
</tbody>
</table>

The von Mises stresses recorded at $B_{COR}$ are plotted against insertion time in Figure 7. The description of the stress characteristics at $B_{COR}$ is detailed in Table 3.

![Figure 7  Stress variations at location $B_{COR}$](image)
### Table 3  Explanation of stress characteristics at $B_{COR}$

<table>
<thead>
<tr>
<th>Insertion depth (mm)</th>
<th>Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>The entire implant torque is only absorbed by the cortical bone. Thus a rapid increase in stress level is evident during this stage.</td>
</tr>
<tr>
<td>1</td>
<td>Stress shows a reduction due to the fact that some stresses are transferred to the cancellous bone. A gradual increase in stress is due to the increase in implant surface area that is in contact with the cortical bone.</td>
</tr>
<tr>
<td>2</td>
<td>The cortical bone is in contact with the interfacial blood and bone. However there is a stress concentration at the tip of the implant thread. Again the stress increases quite rapidly.</td>
</tr>
<tr>
<td>3 - 7</td>
<td>The results are similar to those recorded in the cancellous bone. The torque remains constant between 3 and 7mm insertion that leads to no change in the stress levels for each insertion depth. More surface area contact between the implant and cancellous/cortical bone leads to a step-wise decrease in stress within the cortical bone.</td>
</tr>
<tr>
<td>8 - 9</td>
<td>The increased torque is responsible for the increase in stress levels.</td>
</tr>
<tr>
<td>10</td>
<td>The increase in stress is due to the implant neck being in direct contact with the cortical bone.</td>
</tr>
<tr>
<td>11</td>
<td>At 833Nmm torque, the cancellous and cortical bone shares the stress. Thus a decrease in stress can be seen for both bone. The stress values increase dramatically during this final stage of insertion due to the cortical bone being the strongest and thus carrying the majority of the torque exerted through the implant.</td>
</tr>
</tbody>
</table>

5  **Discussion**

The analysis undertaken in this paper is to replicate the implantation process in a step-wise manner, with the objective of advancing the current understanding of the von Mises stress characteristics within the mandible. The stresses predicted in both cancellous and cortical bone are discussed in some detail. It is found that a stress concentration occurs at the implant neck. This is because a greater resistance is exerted at the implant entrance into the cancellous and cortical bone. The stress characteristics examined in this study offers some insight into the behaviour of the cancellous and cortical bone during the implantation process. Important factors to consider for a successful implantation are the optimum insertion torque and the set time periods for each level of torque applied. The outcome of this study will assist the clinician to perform a patient specific implant treatment in a more quality-controlled manner.

6  **Conclusion**

A simplified and efficient 3D modelling procedure is proposed in this study to examine the von Mises stress characteristics within the mandible during the implantation process. The study takes into account the realistic geometry, material properties, loading and support conditions for both implant and jawbone as well as the biomechanical implant/bone interface. A major modelling assumption made in this study is that the implantation process is simulated in a step-wised manner instead of a continuous process. This assumption is primarily related to the software capabilities. It should be noted that this research is a pilot study aiming to offer an initial understanding of the complicated stress distribution characteristics due to the increasing time-dependant torque.

Future work includes the variation of the torque applied during the implantation process. The magnitude of applied torque and the duration of its application drastically influence the stress level which in turn affects the outcome of implantation. Based on the initial data obtained in this study, more sophisticated software should be used in order to simulate the full dynamical process of implantation.
7 Acknowledgements
This work is made possible by the collaborative efforts between Griffith’s School of Engineering and School of Dentistry and Oral Health. A special thank goes to Mr John Divitini and Mr Fredrik Engman from Neoss Limited, UK for their endless support.

8 References