Dynamic Modelling and Simulation of Dental Implant Insertion Process - A Finite Element Study

HONG GUAN¹, RUDI C. VAN STADEN², NEWELL W. JOHNSON³, YEW-CHAYE LOO⁴

¹Associate Professor in Structural Engineering and Mechanics, Griffith School of Engineering, Griffith University Gold Coast Campus, Queensland, Australia.

²Research Fellow, Griffith School of Engineering, Griffith University Gold Coast Campus, Queensland, Australia.

³Professor of Dental Research, Griffith Health Institute, Griffith University Gold Coast Campus, Queensland, Australia.

⁴Professor and Foundation Chair of Civil Engineering, Director, Internationalisation and Professional Liaison, Science, Environment, Engineering and Technology Group, Griffith University Gold Coast Campus, Queensland, Australia.

January 2011

Address correspondence to Associate Professor Hong Guan. Griffith School of Engineering, Griffith University Gold Coast Campus, Queensland 4222, Australia. Telephone number: +61 (0)7 5552 8708. Fax number: +61 (0)7 5552 8065. Electronic mail: h.guan@griffith.edu.au

Keywords: Dental implant; Implantation procedure; Finite element technique

Subject Classification: Engineering application in dental implant research

GUAN, H., VAN STADEN, R.C., JOHNSON, N.W., LOO, Y.C.

Dynamic Modelling and Simulation of Dental Implant Insertion Process - A Finite Element Study

Finite Element in Analysis and Design

Objectives: Using the finite element technique, the stress characteristics within the mandible are evaluated during a dynamic simulation of the implant insertion process. Implantation scenarios considered are implant thread forming (S1), cutting (S2) and the combination of forming and cutting (S3). Ultimately, the outcome of this study will provide an improved understanding of the failure mechanism consequential to the stress distribution characteristics in the mandible during the implantation process. *Material and Methods:* Parameters considered herein include bone cavity diameters of 3.9mm (for S2), 4.25mm (for S1) and a tapered cavity of diameters linearly varying from 3.9 to 4.25mm (for S3). The bone-implant system is modelled using three-dimensional tetrahedral elements. Idealised bone and implant interaction properties are assumed. The stress profiles in the mandible are examined for all bone cavity diameters. *Results and Conclusion:* The stress levels within the cancellous and cortical bone for S1 are significantly reduced when compared to scenarios S2 and S3. For S3, during the initial insertion steps, the stress is marginally less than that for S2. Close to the end of the insertion process, the stress level within the cancellous bone in S3 is approximately half way between that of S1 and S2. Generally for all scenarios, as the insertion depth increases the stress increases less significantly in the cortical bone than in the cancellous bone. Overall, different implant surface contact areas are the major contributors to the different stress characteristics of each scenario.

1. Introduction

The modern dental implant is a biocompatible device, usually made of titanium, which is surgically placed into a jawbone to support a prosthetic tooth crown to replace a missing tooth or teeth. Worldwide statistics show a high success rate: in excess of 95% retention over a 5 year period if the implants are correctly designed, manufactured and inserted, which is suggested by the studies of Calandriello and Tomatis [4], Gallucci et al. [6] and Lambert et al. [7]. It is strongly believed by implant specialists that a large proportion of the 5% failures are associated with incorrect insertion techniques.

The implantation process is generally initiated by making a small incision into the gingival soft tissue at the proposed implantation site. After the bone is made visible, a pilot hole is drilled using a round bur and only the cortical bone layer is penetrated. Drills of, for example, diameters 2.2, 3.0, 3.6 and 3.9mm may be used successively to make a cavity within the bone for a 4.5mm diameter implant [8]. The implant is finally inserted into the bone cavity manually using a torque ratchet or mechanically using a surgical micro motor. Manual insertion of the implant generally requires an increased torque with insertion depth as a result of the relatively low insertion velocity. On the other hand, mechanical insertion is performed at increased velocity and therefore the torque remains constant regardless of the insertion depth. The entire operation is performed under local anaesthesia and a constant supply of physiological saline during the procedure reduces heat and flushes away blood and bone fragments. The controlled insertion torque and heat help to minimise bone fracture and promote bone healing. The response of the bone in terms of resorption or healing is directly related to the stress within the bone, according to Wolff's theory [15].

During implantation the implant thread either forms, cuts or forms and cuts the surrounding bone depending on the diameter of the bone cavity in relation to the diameter of the implant itself. If the cavity is 0.25mm smaller in diameter than the implant, "thread forming" (S1) takes place. Sennerby and Meredith [13] showed that stability is reduced at sites of S1, thereby increasing the possibility of implant failure. For a bone cavity that is 0.6mm smaller in diameter than the implant, "thread cutting" (S2) occurs. S2 is ideal: the implant cuts a new thread pattern into bone, around a cavity of smaller diameter. In this case optimum implant stability is achieved through the entire length of the implant. Note that if an implant is placed into a cavity created by a newly extracted tooth, the diameter of the cavity will vary. Generally this cavity will be of a larger diameter at the top than the bottom: therefore implant "thread forming and cutting" (S3) takes place during insertion [8]. The amount of forming or cutting ultimately contributes to the biological response of the bone and subsequently the outcome of the implantation procedure.

The implantation process is complex with respect to many biomechanical aspects and a comprehensive representation using the finite element technique is intricate and perhaps computationally inefficient. The purpose of this study is to advance on the work of van Staden et al. [14] by modelling the implantation process in a dynamic manner, including further advancement on the bone-implant contact modelling. Also included in this study are the effects of implant thread forming and cutting on the surrounding bone.

2 Materials and methods

2.1 Three-dimensional representation of bone-implant system

All the modelling and analysis are carried out using ABAQUS [2] Finite Element Analysis (FEA) System. Data acquisition for bone dimensions are based on computed tomography scanned images. From these images a section from the human mandible is taken as illustrated in Figure 1.

Fig. 1. Location of three-dimensional slice in a mandible

Illustrated in Figures 2 are the different dimensions of bone cavities. S1, S2 and S3 take place respectively for bone cavity diameters of 4.25, 3.9mm and the combination of these two. Note that for S3 the bone cavity has a 1.5 degree taperage due to the difference in the top (4.25mm) and bottom (3.9mm) diameters of the cavity, as shown in Figure 2 b). The implant dimensions are shown in Figure 3.

Fig. 2. Details of mandibular bone and cavity.

Fig. 3. Details of a typical implant design.

The finite element models of the bone and implant are automeshed using tetrahedral elements. As an example, for the bone and implant models of S2 the total numbers of elements and nodes are 85234 and 67567 respectively.

Realistic material behaviour of the cancellous and cortical bone is simulated through the definition of elastic and plastic properties, as listed in Table 1. The elastic plastic behaviour of bone is defined by Young's modulus, Poisson's ratio, density, yield stress and plastic strain, as obtained by Burstein et al. [3] from human femur and tibia specimens. The friction coefficient between the surfaces of bone and commercially pure titanium was obtained by Choubey et al. [5] through fretting wear tests performed in a salt solution. The Young's modulus, Poisson's ratio and density of the implant are respectively 102GPa, 0.3 and $4.54 \times 10^{-6} \text{ kg/mm}^3$. The stress-strain relationship for the bone is defined up to the point of fracture, as indicated in Figure 4.

Fig. 4. Stress-strain behaviour [3].

Table 1 Material properties of cancellous and cortical bone.

		Cancellous bone	Cortical bone	Source	Specimen details
	Young's modulus, E ($\times 10^3$ N/mm ²)	0.7	9		
Elastic	Poisson's ratio, ν	0.35	0.3	Burstein et al. [3]	Human femur and tibia
	Density, ρ ($\times 10^{-7}$ kg/mm ³)	5.3	18		
Plastic	Yield stress (N/mm ²)	35	180	(see Figure 4)	
	Plastic strain	0.135	0.015		
Contact	Friction coefficient	0.61		Choubey et al. [5]	Human femur

2.2 Simulation technique

During implantation the implant can be inserted manually or mechanically. A manual insertion process was modelled previously [14] in a step-wise manner with a torque applied to the implant that increases with time. The present study deals with mechanical insertion whereby the process of implantation is continuous with a constant torque.

Figure 5 illustrates a constant torque of 450Nmm with a velocity of 0.31mm/s applied to the top of the implant. The velocity is based on an insertion depth of 11mm over a total period of 36 seconds. Note that the bone cavity is 11.5mm in depth, thereby allowing 0.5mm depth to remain after implant insertion. In a clinical procedure the 0.5mm would allow storage of blood and bone fragments. Shown in Figure 5 are the fixed restraints on the bone surfaces (anterior and posterior) along the mesio-distal direction of a hypothetical human mandible.

Fig. 5. Loading and restraint conditions applied to the bone and implant.

2.3 Stress measurements

The stress within the bone are considered to be the determining factor for understanding both bone fracturing during insertion and subsequent bone resorption. The von Mises stresses are measured along the lines VV in the cancellous bone and HH in the cortical bone, as shown in Figure 6. Line VV is 11.5mm in length for all bone cavity diameters; however, the length of HH is dependant on the bone cavity dimensions, as detailed in Table 2. Due to the irregularity of the mesh, a straight line of nodes at which the stress is measured is only approximated for both VV and HH . The distance of VV away from the bone cavity surface is fixed at 0.5mm. The beginning and end points of VV (i.e. V_1 and V_2) and HH (i.e. H_1 and H_2) are also identified in Figure 6 for ease of discussion.

Table 2 Length of lines VV and HH (in mm).

Line	Thread forming, S1	Thread cutting, S2	Thread forming and cutting, S3
VV	11.5	11.5	11.5
HH	4.4	4.575	4.4

Fig. 6. Stress measurement within cancellous and cortical bone.

2.4 Modelling contact

Interaction between the bone and implant during dynamic simulation of the implantation process is complex and requires definition of contact conditions. In the present study, contact is defined in ABAQUS [2] using “surface-to-surface” discretisation because it provides more accurate stress and pressure results than node-to-surface discretisation. Surface-to-surface contact is incorporated in the modelling by defining the constraint enforcement methods where the surfaces do not require matching meshes (i.e. node to node contact) because ABAQUS [2] enforces conditional constraints on each surface to simulate contact conditions. In addition, the contact interaction properties are also required to be defined for the contact pair.

As illustrated in Figure 7, definition of the two contact surfaces is achieved by setting the side and bottom implant surfaces as the master surface. The slave surfaces include the entire inner surface of the

cavity and the top ring area of 0.5mm width on the cortical surface. This top ring area allows the contact between the implant and the top of cortical bone because the implant diameter is larger than the cavity. In accordance with the surface-to-surface definition, contact constraints are enforced in an average sense over the slave surface, rather than at discrete points, such as at slave nodes in the case of node-to-surface discretisation. Therefore, penetration of individual master nodes into the slave surface may occur, however the reverse is not possible.

Fig. 7. Master and slave surfaces.

Defining the contact is yet to be completed because under large deformations ABAQUS [2] defaults the slave surface to be penetrated by the master nodes, as illustrated in Figure 8 a). As such, the material properties of the slave surface can not be properly defined after deformation. Therefore, adaptive meshing technique must be used so that the material moves with the slave surface mesh at all times during the insertion simulation.

Fig. 8. Adaptive meshing.

The adaptive meshing scheme used in ABAQUS [2] is termed “Arbitrary Lagrangian-Eulerian” (ALE) because it combines the Lagrangian and Eulerian methods. The Lagrangian method is used to track the path of the element so that no nodal penetrations occur. However, this method alone still allows material to move independently of the mesh. The Eulerian method, on the other hand, takes into account conservation of mass so that the material is conserved within the elements. The ALE combines these two methods so that if nodal penetrations occur, forces that are a function of the penetration distance are applied to the master nodes to oppose the penetration, with equal and opposite forces acting on the slave surface at the penetration points. Detailed in Figure 8 b) is an example of the slave surfaces deforming around the master nodes after ALE is applied.

The next step in defining a surface-based contact is to describe the contact interaction properties. In this study the tangential properties of the surface is defined using friction coefficient (see Table 1) and the normal properties defined as hard contact. Hard contact is implemented to ensure that the master nodes are in complete contact with the slave surfaces therefore not allowing transfer of any tensile stresses across the interface.

3 Results and discussion

The von Mises stresses are evaluated within the cancellous and cortical bone for S1, S2 and S3 during the implantation process. Each scenario is discussed below in a separate subsection.

The stresses are measured along the lines VV and HH for eleven implant insertion steps. Figures 9, 11 and 13 presents the stress characteristics within the cancellous bone respectively for S1, S2 and S3. Figures 10, 12 and 14 shows the stress within the cortical bone respectively for S1, S2 and S3. The stress profile within the cancellous and cortical bone for each stage of insertion is presented on the left of the stress contour plot. For ease of discussion time steps of 3.6 seconds are utilised, thereby resulting in ten equally divided time periods for a total of 36 seconds. In addition to this, it has been

assumed that the implant tip is normally pushed slightly into the top surface of cortical bone prior to the application of the torque. This corresponds to 1.8 seconds or 0.5mm insertion depth where the stresses are measured as the first stress profile. This is followed by the profiles corresponding to 3.6 seconds and thereafter at a 3.6 second time step.

3.1 Thread forming, S1

3.1.1 Cancellous bone

For most of the time steps it is evident that when the insertion depth increases the stress level also increases. This is because the surface area of contact between the bone and implant increases and therefore a larger amount of the applied torque is transferred to the bone.

As seen in Figure 9 a), when the implant is inserted 0.5mm into the cortical bone, the stress level within the cancellous bone is relatively low (0.07MPa) because the implant and cancellous bone are not yet in direct contact. At this stage, the only stresses experienced in the cancellous bone are those transferred through the cortical bone. Figure 9 a) also indicates that for insertion depths 1.1 to 2.2mm, the global stress peaks occur at VV_I . These peaks are caused by the primary cutting faces together with the stresses transferred from the cortical bone.

As detailed in Figure 9 b), from the time period 10.8 to 18 seconds the cancellous bone experiences an increase in the stress further away from VV_I because the implant is inserted deeper into the bone. The stress contour illustrated in Figure 9 b) for an insertion depth of 5.5mm indicates a more smoothed-out distribution next to the implant. Figure 9 c) presents the stress profile and contour from 21.6 to 28.8 seconds. In general the stress increases slightly compared to the previous stages (i.e. average from 0.93 to 1.24MPa), again due to the increased bone to implant contact. The stress contour distributes more evenly throughout the bone adjacent to the implant as the insertion depth increases, as illustrated in Figure 9 c).

Figure 9 d) details the stress characteristics at the final stages of insertion. Overall the stress at both 9.9 (1.94MPa) and 11mm (1.48MPa) insertion depths increase compared to all the previous stages. This is again due to the increased contact area between the implant and the cancellous bone. The global stress peaks for these insertion steps are a result of the abrupt change in implant geometry where the implant neck establishes contact with the cancellous bone.

a) 0.5 to 2.2mm

b) 3.3 to 5.5mm

c) 6.6 to 8.8mm

d) 9.9 and 11mm

Fig. 9. Stress characteristics in cancellous bone at each insertion stage during thread forming.

3.1.2 Cortical bone

In general, the stress within the cortical bone decreases from HH_1 towards HH_2 for all insertion steps. Presented in Figure 10 a), for the time period between 1.8 and 7.2 seconds, is a significant stress variation (1.6 to 9.9MPa) that is a result of the primary cutting faces not being in contact with the point HH_1 at 1.8 seconds followed by establishing contact at 3.6 and 7.2 seconds. Such a change in stress is particularly evident in the contour plots in this figure.

For the time period between 10.8 and 18 seconds, as presented in Figure 10 b), the stress next to the implant increases to a range of 14.37 to 23.4MPa. Such an increase in stress is due to the increased cortical bone to implant contact as a result of the narrowing gaps between the cutting faces as the insertion step increases.

As evident in Figure 10 c) for the time period from 21.6 to 28.8 seconds the stress contours are similar to the 5.5mm insertion depth. However, the stress decreases slightly (i.e. from 23.4 to 22.1MPa) because the cutting faces are no longer in contact with the cortical bone at HH_1 . Figure 10 d) illustrates that the stress characteristics for time periods 32.4 and 36 seconds vary little from those of 28.8 seconds. This is because the implant neck is in contact with the cortical bone at this stage instead of at earlier stages where the thread contact causes more abrupt changes in geometry.

c) 6.6 to 8.8mm

d) 9.9 and 11mm

Fig. 10. Stress characteristics in cortical bone at each insertion stage during thread forming.

3.2 Thread cutting, S2

3.2.1 Cancellous bone

Similar to S1, S2 also causes increased stresses when the insertion depth increases. From the time period 1.8 to 7.2 seconds the stresses (0.3, 1.2 and 3.92MPa) are significantly higher than those found during S1 (0.07, 0.51 and 0.96MPa), as illustrated in Figure 11 a). This is because the diameter of the bone cavity is reduced in S2. This also results in a larger stressed region towards the outer edge of the cortical bone, as evident in the stress contour plots. It is also found that stress peaks occur at the same locations as found for S1.

From the time period 10.8 to 18 seconds, as detailed in Figure 11 b), the increased stresses are further down and away from VV_1 due to the increased insertion depth. The stress contours are distributed more unevenly when compared to the previous stages. The stress peaks at VV_1 are much larger (2.5 to 4.8MPa) than those found in S2 (0.8 to 1.08MPa). Figure 11 c) shows the stress profile and contour from 21.6 to 28.8 seconds, in general the stress increases slightly compared to the previous stages.

Figure 11 d) presents the stress characteristics at the final insertion stages. The stress profile and contour are shown to be more evenly distributed as compared to the corresponding ones in S1. The stress peaks occur at 3.8 (5.1MPa) and 4.7mm (5.34MPa) from VV_1 respectively for insertion depths

of 9.9 and 11mm. These stress peaks are also evident for S1 and are a result of the abrupt change in implant geometry where the implant neck commences contact with the cancellous bone.

a) 0.5 to 2.2mm

b) 3.3 to 5.5mm

c) 6.6 to 8.8mm

d) 9.9 and 11mm

Fig. 11. Stress characteristics in cancellous bone at each insertion stage during thread cutting.

3.2.2 Cortical bone

Similar stress characteristics are found for S2 as with S1. However, S2 induces significantly higher stresses within the cortical bone (maximum of 12.1MPa compared to 9.9MPa), as illustrated in Figure 12. At an insertion depth of 5.5mm a maximum stress of 32MPa occurs at the implant neck (Figure 12 b)), which is significantly more than that found at the same insertion step during S1 (23.4MPa) (Figure 10 b)). The stress contours shown in Figures 12 b) to d) also confirms such an increase in stresses where a greater area surrounding the implant is in red.

a) 0.5 to 2.2mm

b) 3.3 to 5.5mm

c) 6.6 to 8.8mm

d) 9.9 and 11mm

Fig. 12. Stress characteristics in cortical bone at each insertion stage during thread cutting.

3.3 Thread forming and cutting, S3

3.3.1 Cancellous bone

As shown previously for S1 and S2, when the insertion depth increases the stresses within the cancellous bone also increase for S3. From the time period 1.8 to 7.2 seconds, as presented in Figure 13 a), the stress level (0.09 to 0.99MPa) is similar to those of S1 (0.07 to 0.96MPa) and smaller than

those for S2 (0.3 to 3.92MPa). This is due to the geometrical changes in the design of the bone cavity. The stress contours are also more comparable to those of S1, as detailed in Figure 13 a).

From the time period 10.8 to 18 seconds, as illustrated in Figure 13 b), the magnitudes of the stresses (1.09 to 1.34MPa) are slightly higher than that of S1 (0.8 to 1.08MPa) and significantly lower than S2 (2.5 to 4.8MPa). Global stress peaks at 14.4 and 18 seconds occur along the line VV because of the geometry of the primary cutting faces and the reducing diameter of the bone cavity which induces a slight change in the stress contour when compared to S1.

Figure 13 c) presents the stress characteristics from 21.6 to 28.8 seconds which shows an increase in magnitude (1.48 to 2.76MPa) when compared to that of S1 (1.09 to 1.46MPa) and decreases compared to S2 (4.57 to 5.24MPa). The stress profiles shown in the figure indicate that stress peaks occur at the region close to the primary cutting faces. A reduction in bone cavity diameter gives a stress contour that is more comparable to that of S1 and S2.

Figure 13 d) shows a significant increase in the stresses, however the stresses at VV_1 remain comparable to that found for S1. The increase in stress at 9.9 (2.32MPa) and 11mm (3.02MPa) insertion depths is a result of the reduced cavity diameter. Note also that the stress contour is again more comparable to S2.

a) 0.5 to 2.2mm

b) 3.3 to 5.5mm

c) 6.6 to 8.8mm

d) 9.9 and 11mm

Fig. 13. Stress characteristics in cancellous bone at each insertion stage during thread forming and cutting.

3.3.2 Cortical bone

The stress characteristics are again comparable to those found for S1 or S2, as detailed in Figure 14. In general the stresses are between those of S1 and S2, but more close to S1 scenario. This is because the stresses are only measured at the top of the bone cavity where the diameter is that of forming.

a) 0.5 to 2.2mm

b) 3.3 to 5.5mm

c) 6.6 to 8.8mm

d) 9.9 and 11mm

Fig. 14. Stress characteristics in cortical bone at each insertion stage during thread forming and cutting.

3.4 Summary of S1, S2 and S3 scenarios

This subsection summarises the maximum von Mises stresses for all insertion steps of S1, S2 and S3 scenarios. Based on data shown in Figures 9 to 14, the maximum stresses along the lines *VV* and *HH* are presented in Table 3. As the insertion depth increases the maximum stress along the line *VV* takes

place at different distances, dv away from VV_I . However for cortical bone the maximum stress always occurs at HH_I .

Insertion depth (mm)	Forming, S1			Cutting, S2			Forming and cutting, S3		
	VV	dv	HH_I	VV	dv	HH_I	VV	dv	HH_I
0.5	0.07	0	1.6	0.3	0	2.1	0.09	0	2.06
1.1	0.51	0	9.2	1.2	0	11.44	0.44	0	9.8
2.2	0.96	0.7	9.9	3.92	0.71	12.1	0.99	0.82	10.9
3.3	0.8	0	14.37	2.5	0	20.3	1.09	1.48	16.1
4.4	1.08	0	14.4	4.02	0.79	20.1	1.34	1.66	16.14
5.5	0.9	1.47	23.4	4.8	0	32	1.11	0.87	26.3
6.6	1.09	0	22.1	4.57	0.97	30.2	1.48	1.59	24.7
7.7	1.46	1.41	16.4	5.2	0	22.6	2.76	5.6	18.4
8.8	1.18	1.65	20.7	5.24	2.75	28.5	2.21	3.91	22.9
9.9	1.94	3.62	17.2	5.1	4.07	22.8	2.32	7.52	19
11	1.48	1.47	20.4	5.34	4.61	28.1	3.02	9.77	22.2

Table 3 reveals that for the cancellous bone, the maximum stresses for S3 are closer in magnitude to those of S1 at initial insertion steps. This is because the upper bone cavity diameters are similar for these two cases. As the insertion depth increases, the maximum stress of the combined scenario approaches a magnitude which is approximately halfway between S1 and S2 scenarios. This is due to the reduction in bone cavity diameter in the lower region which is approaching that of the S2 scenario. In general, the maximum stress along the line VV occurs on or close to VV_I . However, for insertion depths 8.8 to 11mm the maximum stresses occur further away from VV_I due to the decrease in the bone cavity diameter. The maximum stresses at HH_I in the cortical bone for S3 are between those of S1 and S2 from 0.5 to 2.2mm insertion depths. For insertion depths from 3.3 to 11mm the stress levels of S3 are closer in magnitude to S1 than S2, because the cavity diameters at the top of the cortical bone is the same for S2 and S3.

4 Discussion under conclusion

The von Mises stress characteristics within the cancellous and cortical bone are evaluated for thread forming (S1), cutting (S2) and combined forming and cutting (S3). With the definition of both adaptive meshing and contact interaction properties, available in ABAQUS [2], realistic stress characteristics are simulated. The continuous dynamic simulation and implant cutting faces prove to be the major factors that distinguish the present results from those achieved in the step-wise simulation [14].

For S1 the stress levels within the cancellous and cortical bone are less than those in S2 and S3 because of the reduced bone to implant surface contact area. However, the stresses within the cancellous bone are only slightly reduced for S3 during the initial insertion steps. Then at later insertion steps the stress within the cancellous bone (at any location along the line VV) increases to be at a level approximately half way between that of S1 and S2. For cortical bone the magnitude increases less significantly as the implant insertion depth progresses. The minor variation is due to the geometrical differences of the bone cavity. In both cancellous and cortical bone the primary cutting faces induce stress peaks during the initial insertion stages (0.55 to 2.2mm). This is because of the abrupt changes in geometry of the cutting faces. For the final insertion stages (9.9 and 11mm)

the change in implant section (i.e. implant neck establishes contact with the cancellous bone) results in stress peaks within the cancellous bone.

The optimal or desirable stress levels to be experienced by local bone during implantation have not yet been firmly established. However, according to Rieger et al. [11] and O'Mahony et al. [10], this lies between 1.72MPa and 2.76MPa. The material structure of the cancellous bone makes it more sensitive to fracture than the cortical bone. For the purpose of this study the ideal stress range for cancellous bone growth and repair is plotted in Figure 15 together with the minimum and maximum stress profiles along the line VV produced by S1, S2 and S3. Note that the minimum stress profiles are obtained at 1.8 seconds and the maximum at 36 seconds. On the basis of present knowledge, if the stress falls below 1.72MPa bone may not be stimulated adequately for effective healing and osseointegration of the implant. Further, if the stress exceeds 2.76MPa bone resorption may occur, contributing to loosening and potential failure of the implant. Ultimately the stress should remain between these limits.

Fig. 15. Maximum and minimum stress profiles during implantation.

For all three insertion scenarios the minimum stress profiles at 1.8 seconds are considerably below the lower limit of the stimulation stress (1.72MPa). For S1, the maximum stress profile (at 36 sec) still does not reach the lower stress limit of 1.72MPa. The low stress level produced by S1 may adversely affect initial retention of the implant which confirms the findings of Sennerby and Meredith [13] in that thread forming reduces implant stability. For S2, the maximum stress profile at 36 seconds shows that bone resorption may occur around the implant because the upper stress limit of 2.76MPa is exceeded for most of the points along line VV. Overall, S3 best satisfies the ideal stress level suggested in literature because the bone is stimulated with minimum resorption. In current practice many implant companies [1,7,8] generally recommend S2 for normal bone (i.e. type 3 or 4 [12]) and S1 for compact bone (i.e. type 1 or 2) so that implant stability can be compromised. Based on the findings of this study, S3 may also be recommended for clinical practice.

It is important to note, however, that these findings have not modelled the fracture of bone through element deletion, nor the effects of blood flow. Incorporating such aspects into the finite element analysis increases the complexity of the problem and modelling, nevertheless it could be important in determining the true outcome of the implantation process. Our future research will take into account

more detailed contact modelling definition by incorporating aspects such as fracture, blood flow and heat transfer.

5 Acknowledgements

This work is made possible by the collaborative support from Griffith's School of Engineering, School of Dentistry and Oral Health and Griffith Health Institute. A special thank goes to Professor Neil Meredith, Mr John Divitini and Mr Fredrik Engman from Neoss Limited, UK for their endless contributions.

6 References

- [1] 3i Implant Innovations Inc., viewed 13 May 2010, <<http://www.biomet3i.com>>.
- [2] ABAQUS, viewed 5 February 2007, <<http://www.simulia.com>>.
- [3] Burstein, A.H. Reilly, D.T. Martens, M. Aging of bone tissue: mechanical properties, *J. Bone Joint Surg. Am.* 58:1 (1976) 82-86
- [4] Calandriello, R. Tomatis, M. Immediate Occlusal Loading of Single Lower Molars Using Brånemark System(R) Wide Platform TiUnite Implants: A 5-Year Follow-Up Report of a Prospective Clinical Multicenter Study, *Clin. Implant Dent. Relat. Res.* [Epub ahead of print-2009]
- [5] Choubey, A. Basu, B. Balasubramaniam, R. Tribological behaviour of Ti-based alloys in simulated body fluid solution at fretting contacts, *Mat. Sci. Eng.* 379 (2004) 234-239
- [6] Gallucci, G.O. Doughtie, C.B. Hwang, J.W. Fiorellini, J.P. Weber, H.P. Five-year results of fixed implant-supported rehabilitations with distal cantilevers for the edentulous mandible, *Clin. Oral Implants Res.* 20 (2009) 601-607
- [7] Lambert, F.E. Weber, H.P. Susarla, S.M. Belser, U.C. Gallucci, G.O. Descriptive analysis of implant and prosthodontic survival rates with fixed implant-supported rehabilitations in the edentulous maxilla. *J. Periodontol.* 80: 8 (2009) 1220-1230
- [8] Neoss Pty Ltd, Neoss Implant System Surgical Guidelines (2009), Harrogate, United Kingdom.
- [9] Nobel Biocare, viewed 12 July 2010, <<http://www.nobelbiocare.com>>.
- [10] O'Mahony, A. Bowles, Q. Woolsey, G. Robinson, S.J. Spencer, P. Stress distribution in the single-unit osseointegrated dental implant: finite element analyses of axial and off-axial loading, *Implant Dent.* 9: 3 (2000) 207-218
- [11] Rieger, R. Mayberry, M. Brose, M.O. Finite element analysis of six endosseous implants, *J. Prosthet. Dent.* 63: 6 (1990) 671-676
- [12] Rho, J.Y. Ashman, R.B. Turner, C.H. Young's modulus of trabecular and cortical bone material: ultrasonic and microtensile measurements, *J. Biomech.* 26: 2 (1993) 111-119
- [13] Sennerby, L. Meredith, N. Implant stability measurements using resonance frequency analysis: biological and biomechanical aspects and clinical implications, *Periodontol.* 2000. 47 (2008) 51-66
- [14] van Staden, R. Guan, H. Johnson, N.W. Loo, Y.C. Meredith, N. Step-Wise Analysis of Dental Implant Insertion Process Using Finite Element Technique, *Clin. Oral Implants Res.* 19: 3 (2008) 303-313
- [15] Wolff, J. The law of the transformation of the bones (translated from German), Hirschwald, Berlin, 1892