***ISSN:* 1812–1217**

The Effect of Dental Implant Length and Diameter on the Stress Distribution at the Implant-Bone Interface of the Immediate Loading Implants: A 3/D Finite Element Analysis

Talal H Mohamad
BDS, MSc (Asst Prof.)
Dept of Conservative Dentistry
College of Dentistry, University of Mosul, Iraq

Umer Al-Adel
BDS, PhD (Prof.)
Dept of Fixed Prosthodontic
College of Dentistry, University of Damascus, Syria

Eissa Wahbeh
BDS, PhD (Prof.)
Dept of Fixed Prosthodontic
College of Dentistry, University of Damascus, Syria

الخلاصة

الاهدف: لتقييم تأثير طول وقطر الزرعة على قيم وتوزيع الاجهادات لزرعات التحميل الفوري باستخدام تحليل العناصر المنتهية ثلاثي الأبعاد. **المواد وطرائق البحث**:تم تصميم نموذج ثلاثي الأبعاد لزرعة مغروسة في العظم، حيث كانت الزرعة المستخدمة من نوعLEADER -ITALIA/ Fix والمصممة خصيصا للتحميل الفوري. تم استخدام برنامج ANSYS V.12 لبناء النموذج الصلب للزرعة والعظم وإجراء تحليل العناصر المنتهية. تم اختبار طولين (١١ملم و ١٣ملم) وقطرين (٣.٧٥ملم و ٤.٢٥ملم) للزرعة. **النتائج**: أظهرت النتائج أن مناطق الاجهادات العليا لجميع الأطوال والأقطار تقع عند عنق الزرعة، وأن زيادة طول الزرعة من ١١ملم إلى ١٣ملم أدى الى زيادة بسيطة في قيم الاجهاد عند السطح البيني للعظم والزرعة، بينما لم يكن للزيادة في القطر من ٣.٧٥ملم إلى ٤.٢٥ملم أي تأثير ملحوظ في زيادة قيم الاجهاد حول الزرعة السنية. **الاستنتاجات**: إن الزيادة في طول الزرعة قد يؤثر سلبيا على قيم الاجهاد،ولكنه يؤدي إلى توزيع أفضل للاجهاد حول الزرعة، إن الزيادة في قطر الزرعة ليس له تأثير ملحوظ على قيم الاجهاد لكنه أدى إلى تقليل حيز القيم العليا للاجهاد عند عنق الزرعة.

ABSTRACT

Aims: to evaluate the effect of implant length and diameter on the values and distribution of stresses of immediate loading implants by using three-dimensional (3D) finite elements (FE) analysis. **Materials and Method:** A 3-D FE model of an implant embedded in a block of bone was used in this study. The implant was LEADER/ ITALIA-Fix type which is specially designed for immediate loading. ANSYS V.12 program was used to build solid model of the implant and bone, and performing the finite element analysis. Two lengths of implant were used (11mm. and 13mm)., and two diameters (3.75mm. and 4.25mm.). **Results:** The results showed that the areas of maximum stress of all lengths and diameters are located at the neck of implants, the increase in the implant length from 11mm. to 13mm. leading to slightly increase in the stress at the implant-bone interface, while the increase in the diameter from 3.75mm. to 4.25mm. having no significant effect in increasing the value of stresses around dental implants. **Conclusions:** The increase in the implant length is negatively affect on stress values, but it leads to better dissipation of stresses around dental implant. The increase in the diameter having no obvious effect on stress values. However, it reduce the area of maximum stress at the implant neck.
Key wards: immediate loading implants, implant length, implant diameter, finite element analysis.

Mohamad TH, Al-Adel U, Wahbeh E. The Effect of Dental Implant Length and Diameter on the Stress Distribution at the Implant-Bone Interface of the Immediate Loading Implants: A 3/D Finite Element Analysis. *Al–Rafidain Dent J*. 2013; 13(1): 44-51.
Received: 22/5/2011 ***Sent to Referees:*** 30/5/2011 ***Accepted for Publication:*** 18/7/2011

INTRODUCTION

Failures of dental implants in the functional phase of treatment protocol mostly takes place after implant loading which may be manifested as bone resorbtion around the neck of implants.[1,2]

The shape of implants is one of the essential criteria of the resulting biomechanical properties and for osseointegration. Peri-implant stress distribution is of course not solely determined by the design of the implants themselves. The selection of the implants should always take the specific morphology of the patient's bone into consideration.[3,4]

The biomechanical aspects can be related mostly to the implant design (eg, length, diameter, shape and material property) and to the patient physiological condition (eg, bone density, occlusal force and medical condition).[5,6]

It is well known that implant diameter, shape, and load direction influence stress distribution[7]. Although in many cases an increase of the implant diameter allows achieving a more favourable mechanical situation, nevertheless, the largest implant or a maximum diameter does not necessarily yield the optimum clinical outcome.[8]

The finite element method is frequently used to determine stresses in bone around dental implants.[9] By using 3-D finite element analysis, Himmlova *et al* made a mathematical simulation of stress distribution around implants to determine which length and diameter of implants would be best to dissipate stress. The results showed that the maximum stress areas were located around the implant neck. Also, an increase in the implant diameter decreased the maximum von Mises equivalent stress around the implant neck more than an increase in the implant length, as a result of a more favorable distribution of the simulated masticatory forces applied.[10]

Li Lin C. *et al* [5]were introduce a study to evaluate the influence of implant length and bone quality on the biomechanical aspects in alveolar bone and dental implant using non-linear finite element analysis. The simulated results indicated that the variations of cortical bony strains between 13mm and 8mm long implants were not significantly as a result of the same contact areas between implant fixture and cortical bone were found for different implant lengths.

Sertgoz and Guvener [11] used the finite element analysis method to study the effect of cantilever and the implant length on the stress distribution in the fixed partial denture supported implants. They found that the increase in length leading to increase the stress at the implant-bone interface. Reiger et al[12] in his FE study found that the implant length having no obvious effect on stress distribution at the implant-bone interface. The aim of the present study is to evaluate the effect of implant length and diameter on the values and distribution of stresses at the bone – implant interface of immediate loading implants by using three-dimensional (3D) finite elements (FE) analysis.

MATERIALS AND METHOD

A 3/D FE model of an implant embedded in a block of bone was used in this study. The implant was LEADER/ ITALIA-Fix type which is specially designed for immediate loading(one piece fixture and abutment). This type of implant is available with two lengths (11mm. and 13mm.) and two diameters (3.75mm. and 4.25mm) figure (1).

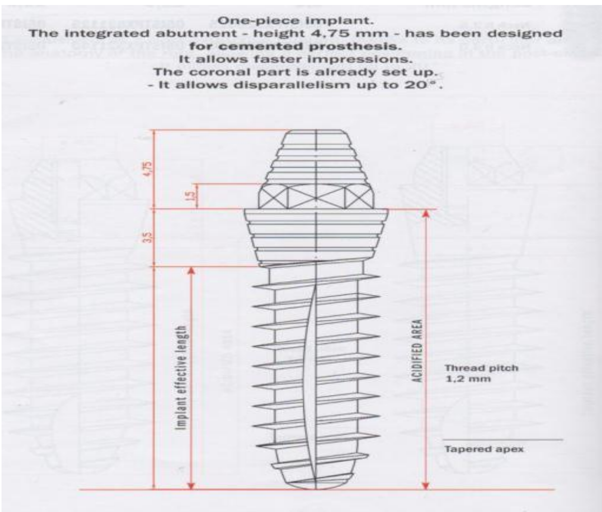

Figure (1): LEADER/ITALIA – Fix implant

The implant was draw in the computer with its real dimensions by AUTO-CAD program (Figure 2).

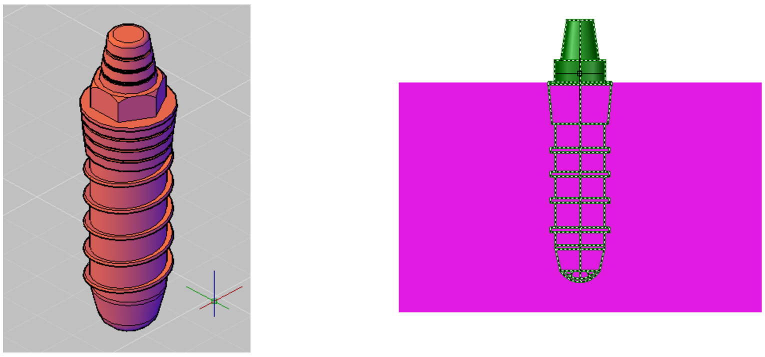

Figure (2): The implant drawn AUTO/CAD program

The 3-D solid finite elements were used to model the bone and implant. The block of bone was 25mm. in height, 15mm. width, and 15mm. length. ANSYS V.12 program was used to build solid model of the implant and bone, and performing the finite element analysis. (Figure 3)

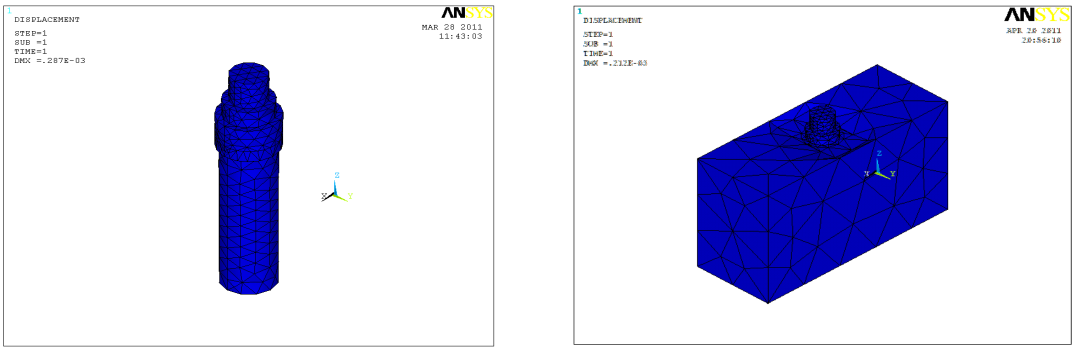

Figure (3): A 3D finite element model of implant and bone block constructed by ANSYS V.12 for FE analysis

Elements: types and description:

The elements used in the model can be described as follows:

1. Titanium implant: SOLID95, 3-D 20-Node Tetrahedral Structural Solid

SOLID95 Element Description:

SOLID95 is a higher order version of the2-D 8-node solid element. It can tolerate irregular shapes without as much loss of accuracy. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries. SOLID95 has plasticity, creep, stress stiffening, large deflection, and large strain capabilities.

2. The bone: SOLID191, 3-D 20-Node Layered Structural Solid.

SOLID191 Element Description:

SOLID191 is a layered version of the 20-node structural solid (SOLID95) designed to model layered thick shells or solids. The element allows up to 100 different material layers. If more than 100 layers are required, the elements may be stacked. SOLID191 has stress stiffening capabilities.

In this study, two lengths of implant were used which are 11mm. and 13mm., and two diameters which are 3.75mm. and 4.25mm. The bone is cortical bone. The Elastic modulus and Possion's ratio of the dental implant and bone used in the study are listed in table (1)

Table (1): Mechanical properties of dental implant and the bone tissue (Li Lin *et al 2006*)

Material	Young Modulus (MPa)	Possion's Ratio
Titanium Implant	110000	0.35
Cortical Bone	14800	0.3

Axially directed force was applied on the top of the abutment to simulate the axial occlusal loading on the implant. The magnitude of force was 2.0 MPa.[13]

RESULTS

To know the values of von Mises stresses and the pattern of their distribution at the implant-bone interface, see the figures of finite element analysis (4),(5),and(7). Each color in the scale represent a value of stress, the red color represent the maximum value of stress, while the dark blue represent the minimum stress value.

● Concerning the effects of the implant length on the values and distribution of stresses at the implant-bone interface (implant diameter was fixed at 3.75mm), the results of the finite elements analysis shows that:

1. The maximum value of von Mises stresses at the implant-bone interface when the implant length was 11mm. is (1.48MPa) at the cervical area of implant, while the minimum stress value is (0.264MPa) at the apex of the implant. (Figure 4, Table 2).

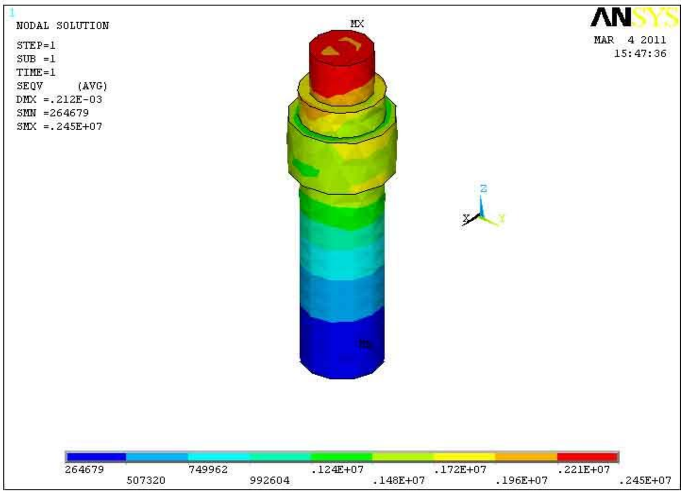

Figure (4): Stress values and distribution along implant-bone interface, the length is 11mm.,the diameter is 3.75mm

Table (2): The maximum and minimum stress values at implant length 11 and 13mm

Length/mm.	Max. value/MPa	Min. value/MPa
11	1.48	0.264
13	1.80	0.401

2. The maximum von Mises stress value when the implant length was 13mm. is (1.80MPa) at the neck of implant, and the minimum stress value is (0.401MPa) at the apical area of the interface. Figure (5), Table (2).

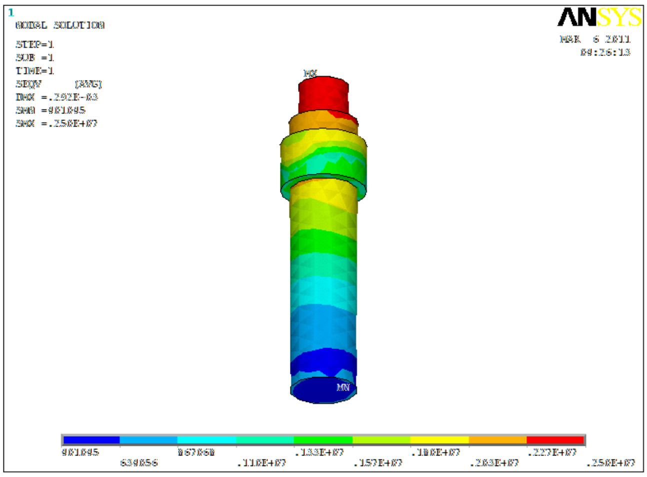

Figure (5): Stress values and distribution along implant-bone interface, the length is 13mm.,the diameter is 3.75mm.

Theses results indicate that the increase in the implant length from 11mm. to 13mm. having a very little effect in increasing the value of stresses around dental implants Figure(6).

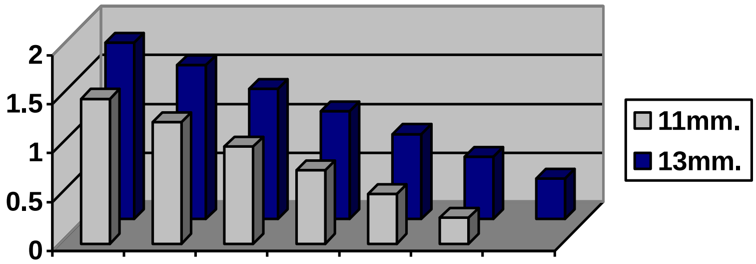

Figure (6): Histogram showing the values of stresses with implant length 11mm. and 13mm.and implant diameter 3.75mm.

● The results of the effects of the diameter on the values and distribution of stresses at the implant-bone interface(the implant length was fixed at 13mm.) shows that:

1. The maximum and minimum von Mises stresses when the implant diameter was 3.75mm. were (1.80MPa) at the neck of implant and the minimum value (0.401MPa). (Figure 5, Table3).

Table(3): The maximum and minimum stress values at implant diameter 3.75 and 4.25mm.

Diameter/mm.	Max. value/MPa	Min. value/MPa
3.75	1.80	0.401
4.25	1.81	0.412

2. When the implant diameter was 4.25mm, the maximum value of stress was (1.81mm.) at the cervical area of implant while the minimum value of stress was (0.412MPa) at the apex of the implant. (Figure7, Table 3).

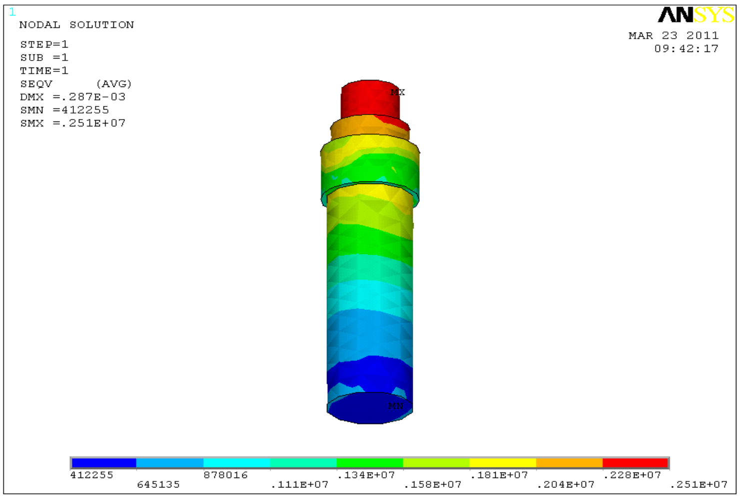

Figure (7): Stress values and distribution along implant-bone interface, the length is 13mm.,the diameter is 4.25mm.

These results reveal that the increase in the implant diameter from 3.75mm. to 4.25mm. having no significant effect in increasing the value of stresses around dental implants. (Figure 8)

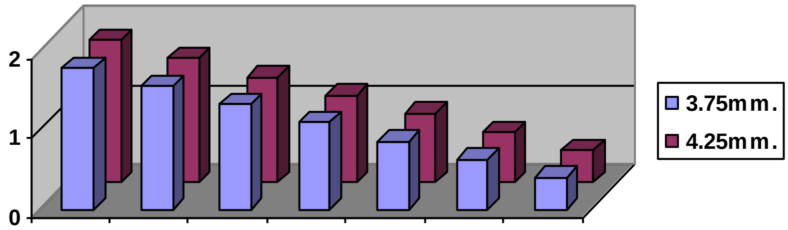

Figure (8): Histogram showing the values of stresses with implant diameter 3.75mm. and 4.25mm.

DISCUSSION

The finite element method is one of the most frequently used methods in stress analysis in both industry and science. It is used for analyzing dental implants. The results of the FEA computation depend on many individual factors, including material properties, boundary conditions, interface definition, and also on the overall approach to the model. It is apparent that the presented model was only an approximation of the clinical situation.[10,15]The model simplification, for example, the implant in the shape of a cylinder rather than a screw or other shapes commonly used in clinical practice, and the simplification of material properties (the entire volume of bone was homogeneous, isotropic with the character of cortical bone) made it possible to reduce the required computer time without affecting the purpose of the study,[10] which was to establish the relative importance of the implant length and diameter.

In this study, two lengths (11mm. and 13mm.) and two diameters (3.75mm. and 4.25mm.) of implant were used for analysis because the company which construct this system introduced only these two lengths and diameters.

An implant- bone model was developed to evaluate the effect of different implant length and diameter on the values and distribution of stresses at the implant-bone interface of the immediate loading implants by means of finite elements analysis.

The Figures of FE analysis (4,5,and7) shows that the areas of maximum stress of all lengths and diameters are located at the neck of dental implants. This result come in agreement to the results of Murad and Al-Adel,[13] and Himmlova et al.[10]

The implant length: when comparing the maximum value of von Mises stresses of the two implant lengths, the results showed that the increase in the implant length from 11mm. to 13mm. leading to slightly increase in the stress at the implant-bone interface. However, 13mm. long implant leading to better dissipation of stresses around dental implant than 11mm. Figure (4,5). This result comes in agreement with Sertgoz and Guvener [11] who found that the increase in length leading to increase the stress at the implant-bone interface. Reiger et al.[12] who found that the implant length having no clear effect on stress distribution. However, Himmlova et al [10] concluded in his study that increasing implant length having a little effect in reducing stresses.

3. The implant diameter: the results of the stress analysis at the implant-bone interface of the two implant diameters showed that there was no difference in the stress values between 3.75mm. and 4.25mm. diameters. This may be due the difference between the two diameters was very small (0.5mm). However, when analyzing the stress distribution, it was very clear that increasing diameter to 4.25mm. led to reduction in the area of maximum stress value at the neck of implant in comparison to that of 3.75mm. diameter,(i.e) the area of maximum stress was wider with implant diameter 3.75mm than the area of maximum stress with diameter 4.25mm. Figure (5,7), this result coincides with the results of ، Himmlova et al [10]. who concluded that the area of high stress at the neck of implant decrease with increasing implant diameter.

CONCLUSIONS

Within the limitations of this FE study the following conclusions can be extracted:

The increase in the implant length is negatively affect on stress values at the implant-bone interface. on the other hand, it leads to better dissipation of stresses around dental implant.

The increase in the implant diameter having no obvious effect on stress values at the implant-bone interface. However, it reduces the area of maximum stress at the implant neck.

REFERENCES

1. Wiskott HW, Belser UC. Lack of integration of smooth titanium surfaces: a working hypothesis based on strains generated in the surrounding bone. *Clin Oral Implants Res* 1999; 10: 429-44.
2. Sethi A, Kaus T, Sochor P. The use of angulated abutments in implant dentistry: five year clinical results of an ongoing prospective study. *Int J Oral Maxillofac Implants*. 2000;15: 801-10.
3. Joos U, Vollmer D and Kleinheinz J. Effect of implant geometry on strain distribution in peri-implant bone. *Mund Kiefer Gesichtschir*. 2000; 4: 143–147.
4. Chun HJ, Cheong, SY, Han, JH, Heo SJ, Chung JP, Rhyu IC, Choi YC, Baik HK, Ku Y and Kim MH. Evaluation of design parameters of osseointegrated dental implants using finite element analysis. *J Oral Rehab* 2002;n29: 565–574.
5. Li lin C, Chan kuo Y, Sheng lin T. Effects of dental implant length and bone quality on biomechanical responses in bone around implants: A 3-D non-linear finite element analysis. *nBiomechanical engineering-applications, bases, and communications* 2005; 17(1): 44-49.
6. Lisa A, Kang B, Wang R, Brien R. Finite element analysis to determine implant preload. *J Prosthet Dent.* 2003; 90: 539-46.
7. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. Evaluating parameters of osseointegrated dental implants using finite element analysis—a two dimensional comparative study examining the effects of implant diameter, implant shape, and load direction. *J Oral Implantol*. 1998; 24: 80-8.
8. Schicho K, Kastner J, Klingesberger R,Seemann R, Enislidis G, Undt G,Wanschitz F, Figl M, Wagner A, Ewers R. Surface area analysis of dental implants using micro-computed tomography. *Clin Oral Impl Res.* 2007; 18: 459–464.
9. Chaichanasiri E, Nanakorn P, Tharanon W, Sloten J. Finite Element Analysis of Bone around a Dental Implant Supporting a Crown with a Premature

Contact. *J Med Assoc Thai*. 2009; 92 (10): 1336-44.
10. Himmlova´ L, Dosta´lova´ T, Ka´covsky´ A, Svatava´ K. Influence of implant length and diameter on stress distribution: A finite element analysis. *J Prosthet Dent*. 2004; 91: 20-5.
11. Sertgoz A, Gurvener S. Finite element analysis of the effect of cantilever and implant length on stress distribution in an implant supported fixed prosthesis. *J Prosthet Dent* 1996;76:165-169.
12. Reiger M, Adams W, Kinzel G, Brose M, Finite element analysis of bone-adapted and bone-bonded endosseous implants. *J Prosthet Dent* .1989; 62: 436-440.
13. Murad ML, Al-Adel U. Study of the effect of prosthesis axis inclination on implant success. Ph.D. thesis, university of Damascus- College of Dentistry. 2006; 77-79.
14. Li Lin C. et al. Finite element analysis of biomechanical interactions of a tooth-implant splinting system for various bone qualities. *Chang Gung Med J*. 2006; 29:143-53.
15. Lai H, Zhang F, Zhang B, Yang C, Xue M. Influence of percentage of osseointegration on stress distribution around dental implants. *Chin J Dent Res*. 1998; 1: 7-11.