ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 6, June 2013

"A REVIEW ON APPLICATION OF CAD AND FEM TECHNOLOGY IN DESIGN OF TAPER DENTAL IMPLANT"

Dhiraj Agnihotri[1] , Amol More[2] , Dr. P.G.Patil[3]

M.Tech CAD/CAM, Department of Mechanical Engineering, G.H.R.C.E, , Nagpur, Maharashtra, India[1]

Assistant Professor, Department of Mechanical Engineering, G.H.R.C.E, Nagpur, Maharashtra, India[2]

Assistant Professor, Department of Prosthodontics, Government Dental College and Hospital, Nagpur, Maharashtra,India[3]

Abstract: This paper provides a detailed review on previous study of design of taper implant with the help of finite element method (FEM).A comparative study between cylindrical and taper implants is done so as to obtain best feasible design of implant with a view to provide minimum stress distribution around surrounding bone. Various aspects such as implant thread profiles, material properties of implant, and other factors that reduce stress distribution are discussed so as to fulfil the gap between different studies.

Keywords: CAD and FEM, dental implant, Osseo integration, crestal bone area.

I. Introduction

CAD and FEM environment provides various advantages of modelling and analysis over other conventional methods. Hence it can be used in dental industry for its data accuracy and data interpretational capability. Dental implant is used to provide support to the artificial tooth and it is a screw like fixture surgically placed into human jawbone. Abutment is screwed into implant to fix artificial tooth on it. Fig.1 shows the basic assembly of these parts in human jaw [1].Since from last couple of decades various researches have been done on dental implant; it leads to development of various types of advanced dental implants [2]. The success of dental implant mainly depends on its biomechanical bonding with surrounding bone; it is also called as Osseo integration phenomenon in dental field. Other factors such as taper of implant, thread profile, material of implant etc. affect the Osseo integration. It is obvious that if bone-implant surface area is increased then there will be good overall distribution of stress around surrounding region [3]. Hence it is important to study stress distribution of various implants having certain range of taper angles and various thread profiles so that to obtain best design of implant.

II. Design and Modelling of Implant

Design of implant involves the study of various aspects such as overall geometry of implant, taper of implant, length and diameter of implant, types of threads etc. so that stress is uniformly distributed around implant surface.

A. *LENGTH AND DIAMETER OF IMPLANT*

Pierrisnard et al. found in their study that the stress to which implants were exposed increased as the length of the implant increased (range, 6 to 12 mm) while the maximum bone stress was found to be almost constant[4]. Study of Petrie CS et al came to the conclusion that strain near crestal bone area reduces nearly by 300% due to increase in diameter as compared to the 165% reduction due to increase in length of implant[5]. Therefore comparatively larger diameter of implant design is preferred.

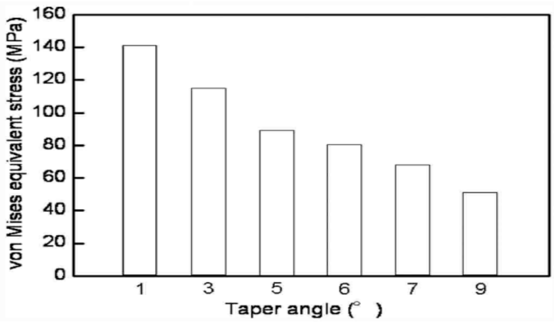
ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 6, June 2013

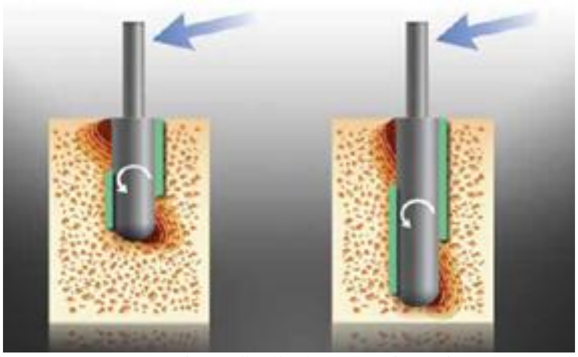
B. *TAPER ANGLE OF IMPLANT*

Taper angle of implant is one of the important considerations in stress distribution around implant area. Tsuyoshi kitagawa et al found in their study that equivalent von mises stress distribution decreases with increase in taper angle of implant. Fig.2 shows that minimum stress occurs for implant having maximum taper angle. But Maitath et al reported that implant having zero taper angle i.e. cylindrical implant produces more desirable stress profile around implant surface than conical implant having certain degree of taper angle [7]. Survival rate of implant inserted in soft quality bone is less than that of implant inserted in good quality bone. Hence there is a need for taper implant [8].

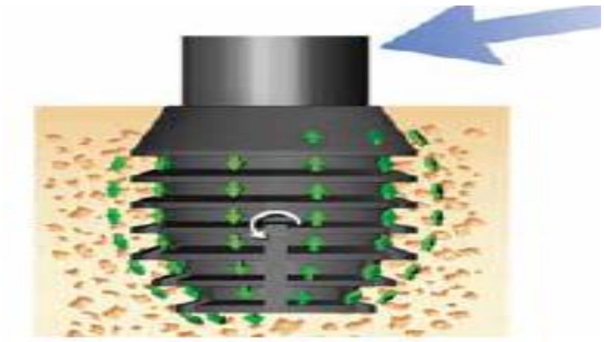
"Fig. 1 Showing implant, abutment and tooth" [1]



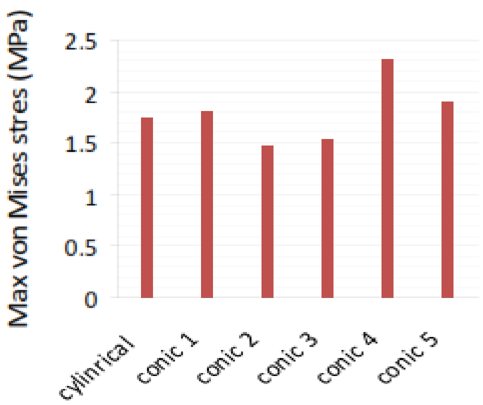
"Fig.2 Von mises stress v/s various taper angle of implant" [6]



"Fig.3 Areas of distribution of forces around cylindrical shaped implants" [9]



"Fig.4 Areas of distribution of forces around plateau (fin) shaped/taper implant" [9]



Implant type	Degree of taper	Max von Mises stress (Mpa)	Max von Mises strain (10^{-4})
Cylindrical	0.00	1.75	0.154
Conic 1	0.02	1.80	0.165
Conic 2	0.06	1.46	0.129
Conic 3	0.10	1.54	0.135
Conic 4	0.12	2.30	0.202
Conic 5	0.16	1.91	0.168

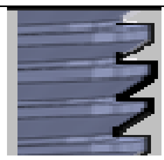
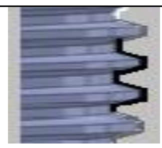
"Fig.5 Maximum von mises stress" [10]

"Table 1 Maximum von Mises stress and strain for all types of implants under 30 psi axial pressure"[10]

Fig.3 and Fig.4 shows the areas of distribution of forces around cylindrical shaped implants and areas of distribution of forces around plateau (fin) shaped implants respectively [9].In cylindrical shape implant unstressed (wasted) implant area is shown in fig.3 by green bar and it is area over which very small amount of stress is distributed and therefore more stressed is on upper and lower part of implant. Distribution of forces around implant mainly occurs in specific area of its length and middle portion of implant is wasted with respect to force distribution. Therefore more area should be provided to that part of implant where force is more distributed as compared to other area where less distribution occurs[9].Taper implant fulfill this criteria by providing taper to its outer surface, hence force is more uniformly distributed as shown in fig.4.
Heidari et al compares stress pattern obtained in implant with zero taper angle (cylindrical implant) with implant having different taper angle and found that cylindrical screw implant generated the lowest maximum von Mises stress in cortical bone and tapered implant with highest tapering generated the highest maximum von Mises stress, probably because of reduced and sharper surface area [10]. From fig.5 conic 4 type of implant with taper 0.12^0 produces highest von mises stress. Other results are shown in Table 1. Results obtained by Heidari et al [10] are quite opposite to those by Tsuyoshi kitagawa et al [6]. It may be due to different taper angle for implant or due to different boundary conditions or due to various assumptions made by these authors during modelling of various cylindrical as well as taper dental implants.

C. *THREAD PROFILE OF IMPLANT*

Implant is threaded to maximize the initial contact, enhance surface area and facilitate dissipation of loads at the body implant interface [11]. There are three main parameters of thread –thread pitch, thread shape, and thread depth. The initial contact and surface area of implant to bone interface will affect the ability of osseointegration [12].

Type of thread	
Buttress	
Reverse Buttress	
V-shape	

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 6, June 2013

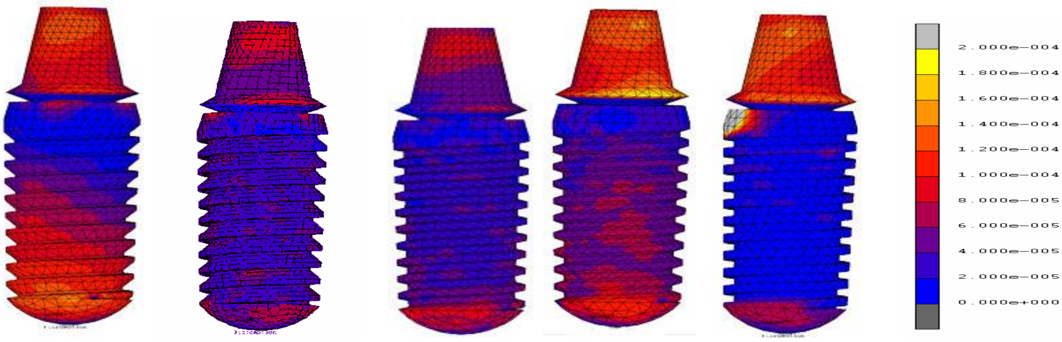
Sinusoidal	
Square	

"Table 2 Types of thread" [12]

Different profile of thread- buttress, reverse buttress, V-shape, sinusoidal and square are shown in Table 2. The edge of thread transfers stress to the bone when implant is loaded [11]. A buttress or square thread profile would transmit compressive forces to the bone under axial load [13]. Aishah Shafi et al perform FEM analysis on implants having various thread profile and compare them with respect to displacement of dental implant system and micro motion along bone- implant system is evaluated. In their study they found that buttress, square and sinusoidal thread profile produce high micro motion at the abutment.

This predicted the less ability of Osseo integration around these kinds of threads [12]. Again less micro motion was represented at the reverse buttress and V-shape thread. However, the V-shape thread produces high deformation after loading. This situation demonstrates that there is more bone loss at the V-shape thread dental implant. This statement proved by Zechner et al. (2004) reveal that bone loss round V-shape thread is higher that square thread via clinical report. Fig. 6 reveals the micro motion of dental implant system obtained by Aishah Ahmad Shafi et al [12].

From the above finding it can be concluded that implant with reverse buttress can provide better Osseo integration property which is very important factor for implant success [12].



a) Buttress b) Reverse buttress c) V-shape d) Sinusoidal e) Square

"Fig.6 Micro motion of dental implants system" [12]

III. FINITE ELEMENT METHOD OF IMPLANT ASSEMBLY

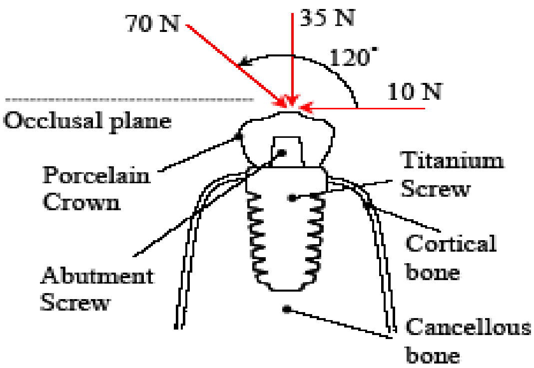
ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 6, June 2013

In most of the previous study, two dimensional models (2D) of implant-abutment system are used. Hence they are failed to simulate actual condition of system. Therefore the results that they got are far from reality and cannot be used as guidance for further study. Three dimensional models (3D) of implant-abutment system more closely simulate actual condition of dental implant system and the stress in the jawbone predicted by a 2D model is less accurate than that predicted by its 3D counterpart [14].

A. *LOADING IN FEM*

Loading conditions of dental implant is one of the most difficult parts of FEM analysis which are meant to reproduce various loads acting in human jaw bone during various working conditions of jaw bone. Hence it is important to simulate more accurate loading conditions in FEM analysis for getting more accurate results.



"Fig.7 loading on implant abutment system" [15, 16, 17]

Generally most of the researchers assume that three types of loads i.e. horizontal, vertical and oblique are acting in human jaw during chewing operation [15, 16, 17]. Out of them Canay et al [15] and Tada et al [16] both assumed a horizontal load of 50N and a vertical load of 100N with no oblique forces in their implant model whereas Meijer et al [17] assumed a horizontal force of 10N, vertical force of 35N and an oblique force of 70N at an angle of 120 degrees from the horizontal, as shown in Fig.7.

B. *MATERIAL PROPERTIES IN FEM*

Homogeneous, linear, elastic material behaviour for the jawbone is assumed in FEM in most of the cases for simplicity which is characterized by a single Young's modulus and Poisson's ratio [18, 19, 20]. Titanium material is used for implant in most of studies

IV. RECOMMENDATIONS FOR FUTURE RESEARCH DIRECTIONS

Due to availability of various design and FEM software's, it is quite easy to model the bone-implant interface and undergo analysis. But due to involvement of various parameters like thread

ISSN: 2319-8753

International Journal of Innovative Research in Science, Engineering and Technology
Vol. 2, Issue 6, June 2013

profiles, taper angle, boundary condition, material properties of bone-implant interface etc. is very difficult to simulate actual condition of implant in human jaw bone with considerable accuracy.

Hence detailed study is required for getting more accurate interception of implant-bone model which is more close to actual condition and more reliable. Again design of a particular implant is suitable for a particular patient but may not be suitable for other because properties of bone may vary from patient to patient and therefore certain method must be developed to obtain properties of implant-bone interface of individual patient and then only those implants are selected whose properties are more compatible to the jaw bone properties of patient. Previous studies provide the evidence that more concentration is given to individual parameter i.e. either on thread profile or taper of implant or other [6, 10, 12]. Therefore more studies considering all such parameters collectively are required to move closer to reality.

V. CONCLUSIONS

Finite element analysis (FEA) has been used more dominantly to analyze biomechanical behaviour of implant in human jaw bone model. Various assumptions are made during FEA in various studies are quite different from each other because no standard data is available. More concentration should be given on bone-implant interface properties.

More correct model of implant-bone system which is closer to actual situation is needed to be used during analysis so that to provide more accurate results and to have more belief of dentist on result obtained in FEM.

REFERENCES

[1] www.biohorizons.com/services/taper implant

[2] Mansour Rismanchian, Reza Birang, Mahdi Shahmoradi, Hassan Talebi, Reza Jabar Zare: Developing a New Dental Implant Design and Comparing its Biomechanical Features with Four Designs. Dental Research Journal (Vol. 7, No. 2, Summer -Autumn 2010)

[3] Himmlova L, Dostalova T, Kacovsky A, Konvickova S: Influence of implant length and diameter on stress distribution: finite element analysis. J Prosthetic Dent.2004; 91(1):20-25.

[4] Pierrisnard L, Renouard F, Renault P, Barquins M: Influence of Implant length and bicortical anchorage on implant stress distribution. Clin Implant Dent Relat Res.2003; 5(4):254-262.

[5] Petrie CS, Williams JL. Comparative evaluation of implant designs: influence of diameter, length, and taper on strains in the alveolar crest: a three-dimensional finite-element analysis. Clin Oral Implants Res.2005Aug; 16(4):486-494.

[6] Kitagawa T, Tanimoto Y, Nishiyama N, Aida M: Application of finite element analysis for taper implant-abutment joints in dental implant system. Int j oral-med sci 7(1):1-6, 2008

[7] Mailath, G., Stoiber, B., Watzek, G., and Matejka, M., 1989, Boneresorption at the entry of osseointegrated implants--a biomechanical phenomenon. Finite element study. Journal of Oral Implantol, 24(2), 80-88.

[8] Astrand, P., Billstrom, C., Feldmann, H., Fischer, K., Henricsson, V., Johansson, B., Nystrom, E., and Sunzel, B., 2003, Tapered implants in jaws with soft bone quality: a clinical and radiographic 1-year study of the Branemark System Mark IV fixture. Clinical ImplantDentistry Related Research, 5(4), 213-218.

[9] Pullen G,Short Implant: Reality and Predictability. A peer Review Publication Heidari B, Bisadi H, Heidari Be, Kadkhodazadeh M. Influence of Different Tapered Implants on Stress and Strain Distribution in Bone and Implant: A Finite Element Analysis. J Periodontal Implant Dent 2009; 1(1):11-19 (2009e2)

[10] Heidari B, Bisadi H, Heidari Be, Kadkhodazadeh M. Influence of Different Tapered Implants on Stress and Strain Distribution in Bone and Implant: A Finite Element Analysis. J Periodontal Implant Dent 2009; 1(1):11-19 (2009e2)

[11] Misch, C.E., Scientific Rationale for Dental Implant Design, in Contemporary implant dentistry, J.T.S. Carl E. Misch, Martha Warren Bidez, Editor. 2008, Mosby Elsevier: Canada.

[12] Shafi A, Kadir M, Sulaiman E: Simulation on the Effects of Dental Implant Threads to the Osseointegration Potential. IEEE Conference on Biomedical Engineering & Sciences. 2010 Kuala Lumpur, Malaysia.

[13] Carl E.Misch, J.T.S., Martha Warren Bidez, Scientific Rationale for Dental Impalnt Design, in Contemporary implant dentistry, C. E.Misch, Editor. 2008, Mosby Elsevier: Canada. p. 203-216.

[14] DeTolla, D.H., Andreana, S., Patra, A., Buhite, R., and Comella, B., 2000, Role of the finite element model in dental implants. Journal of Oral Implantol, 26(2), 77-81.

[15] Canay, S., Hersek, N., Akpinar, I., and Asik, Z., 1996, Comparison of stress distribution around vertical and angled implants with finite-element analysis. Quintessence International, 27(9), 591-598

[16] Tada, S., Stegaroiu, R., Kitamura, E., Miyakawa, O., and Kusakari, H., 2003, Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. International Journal of Oral Maxillofacial Implants, 18(3), 357-368.

[17] Meijer, H.J., Starmans, F.J., Steen, W.H., and Bosman, F., 1993, A three-dimensional, finite-element analysis of bone around dental implants in an edentulous human mandible. Archives of Oral Biology, 38(6), 491-496.

[18] Lewinstein, I., Banks-Sills, L., and Eliasi, R., 1995, A finite element analysis of a new system (IL) for supporting an implant-retained cantilever prosthesis. International Journal of Oral Maxillofacial Implants, 10(3), 355-366.

[19] Mihalko, W.M., May, T.C., Kay, J.F., and Krause, W.R., 1992, Finite element analysis of interface geometry effects on the crestal bone surrounding a dental implant. Journal of Implant Dentistry, 1(3), 212-217.

[20] Nishihara, K., and Nakagiri, S., 1994, Biomechanical studies on newly tailored artificial dental root. Biomedical Material Engineering, 4(3), 141-149.