A Review: Design of Dental Implant Abutment by CAD and FEM Analysis

Md Maroof Md Nayeem[1] Adnan Iqbal Shaikh[2] Ashfaq Jamkhandikar[3] Akib Nisar Dakhwe[4] Salman Jamadar[5]
[1,2,3,4,5]Department of Mechanical Engineering
[1,2,3,4,5]A.R. Kalsekar Polytechnic, New Panvel, Maharashtra, India

Abstract— The purpose of this article is to provide a review of the achievements and advancements in dental technology brought about by computer-aided design and the all-powerful finite element method of analysis. Failures of implant–abutment connections are relatively frequent clinical problems. So there is a need of analysis of dental implant abutment. For that research has been done on the existing design with its limitation. Screw loosening is considered to be a common problem with both screw-retained and cemented implant restorations.1 several complications may arise as a result of loose retaining or abutment screws. Loose screws are more apt to fracture under load, leading to long-term prosthesis complications. This paper focuses on Analysis and techniques used for optimization of Dental Implant.

Key words: Dental Implant, Design Parameter, Optimization, Analysis, Osseo Integration

I. Introduction

Implants could be considered predictable tools for replacing missing teeth or teeth that are irrational to treat. Today, implant success is evaluated from the esthetic and mechanical perspectives. Both depend on the degree and integrity of the bond created between the implant and the surrounding bone. Finite elemental analysis (FEA) is an efficient technique for investigating biomechanical interactions of different implant designs. Numerous studies have been performed to assess force distribution following load application to implants of various dimensions. To enhance clinical success, it is necessary to under-stand how the stress concentration on implants is affected by the shape, width, and height of thread. The use of the finite element method (FEM) in implant biomechanics analysis offers many advantages over other methods in simulating the complexity of clinical situations. 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.

II. Problem Statement

“To study the existing dental implant design and to find the drawback in design. Select optimize technique for Dental Implant analysis.”

III. Design and Modelling of Implant

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

IV. Optimization Process

The most likely cause of the majority of screw loosening is inadequate tightening of the screw. Another important factor is the design and nature (design refers to shape, thread style, head design, and driver shape needed to insert, while nature refers to type of metal) of the screw itself. It was discovered that internally hexed screws could be tightened (even by hand) to a higher degree than slotted screws. When a screw is tightened, a tensile force (preload) is built up in the stem of the screw. This preload creates a contact between the abutment and implant. The closer the tightening force approaches the recommended force for any particular screw, the more stable the connection will be. Thus, the design of the head and body of the screw is significant and should allow a maximum of torque to be introduced in the stem of the screw. The design of the screw head, screw material and tightening force are all important parameters for screw joint stability.[1]

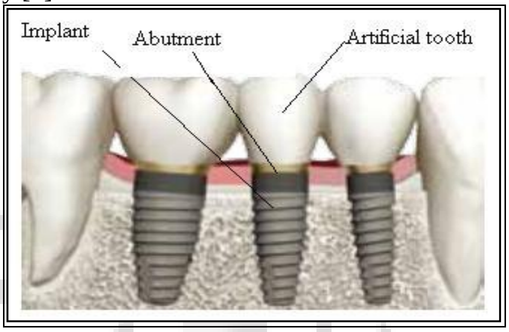

Fig. 1: Showing implant, abutment and tooth.

Most dental implant systems consist of two components: the endosteal part (implant), which is placed in a first surgical phase, and a trans mucosal connection (abutment), which is typically attached after successful implant osseointegration to support the prosthetic restoration. During chewing and biting, the prosthetic restoration and the implant abutment connection is affected by various physiological forces. Dynamic loading in a chewing simulator was used to simulate clinical loading conditions that might lead to a failure of the implant–abutment connection. Under technical considerations, testing the fracture strength of the implant– abutment connection is not possible in a 100% axial direction. When there is an angulated load direction the testing machine was able to detect the bending and the subsequent abrupt loss of tension when the implant–abutment connection failed. [2]

The development of a new prosthesis designed to treat problem through new fixation geometry was based on the static loading and finite element analysis. To understand the behaviour of stress and its distribution along the implant, different types of implant, threaded cylindrical implant and spherical-lobe implant (solid-lobar implant) and hollow-lobar implant are analysed. The ANSYS Finite Element Package was used. The resulting 3-D model volumes were discredited using 8-node explicit brick elements. The bone-implant interface was considered fully bonded. Axis symmetric FE models are constructed for all implant-abutment-bone systems. The implants and abutments are modelled as Ti6Al4V with linear-elastic, isotropic and homogenous properties. Based on analysis and discarding the models with higher values of implant stresses, entering the new design parameter shows a good compromise between the stress shielding and mechanical performance of the dental implant prosthesis according to the finite element analysis results.[3]

Computer-aided design (CAD) software was used to construct a model of the bone block based on a cross-sectional image of the human mandible in the molar region. The implant with a length of 10 mm and a width of 3.75 mm was constructed by using CAD software. After obtaining all the models, solid models were exported commercial FE software to generate the FE models. A 2-D finite element model of an implant–bone system is developed by using ANSYS. Plane strain analysis is used for structures. [4]

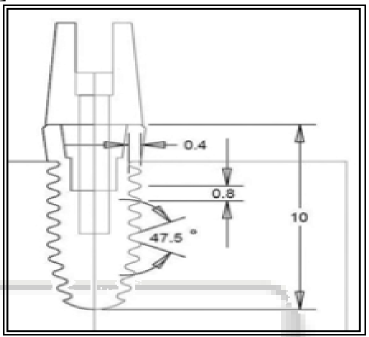

Fig. 2: Dental implant models Type-A.

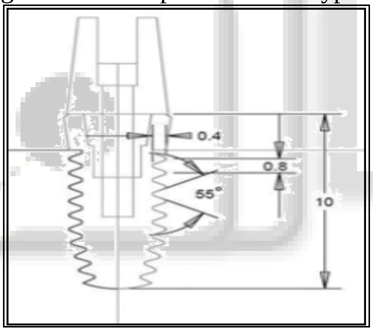

Fig. 3: Dental implant models Type-B.

Implant abutments and frameworks are required to fulfill biological, functional, and esthetic demands. For this reason, the implant abutments and/or frameworks should be made from biocompatible materials with adequate mechanical properties. Even more, they should accurately and passively fit on their mating implants to prevent complications such as screw loosening, bone loss, and abutment fracture during function. The objective of this systematic review was to evaluate the existing scientific evidence in human clinical studies describing the application of CAD/CAM technology in restorative implant dentistry. These applications include CAD/CAM-fabricated abutments and/or frameworks. [5]

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. It is obvious that if bone-implant surface area is increased then there will be good overall distribution of stress around surrounding region. 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. [6]

With the rapid development of computer technology, computer- aided design/computer-aided manufacturing (CAD/CAM) custom abutments have been gradually used in some dental implant systems. In comparison to cast abutments, there is no need for complicated and time-consuming titanium casting and post-casting manipulation that may produce inaccuracies; hence, the CAD/CAM abutment fit may be more precise. Moreover, the CAD/ CAM custom abutments have better physical properties since the material is processed from a homogenous mass under more controlled conditions, with no need for abutment inventory. [9]

A literature review of clinical complications of osseointegrated implants showed that screw loosening or screw fracture varied between 2% and 45% of the implant restorations, with the highest amount in single crowns. A recently published meta-analysis on implant- related complications calculated a cumulative incidence of connection-related complications (screw loosening or fracture) of 7.3% after 5 years of clinical service. Unfortunately, implant companies mostly do not provide specific data on such connection-related complications for their systems with their specific implant–abutment connection design. [10]

A large amount of research has been carried out to determine which factors influence the preservation or loss of the bone around implants, which has helped to achieve the high rates mentioned and reduce the risk of implant failure as a result of peri-implant bone loss. Platform switching reduced the stress values on the abutment and retention screw of single-unit prosthesis during oblique loading. A study carried out how the stress is distributed lengthwise by the abutment up to the bone-implant contact in direction of the applied force.[12]

V. Finite Element Modelling

In order to validate the structure of this custom abutment and to study whether it would be beneficial to the stability of bone and implant, finite element method was used to evaluate the biomechanical performance of the whole implant system in the surrounding bone. The cortical bone was modelled with a layer thickness of about1.5mm around the implant. In this study the mechanical properties of implant, abutment and screw were all treated to be isotropic, homogeneous and linear elastic. And titanium alloy (Ti–6Al–4V) with Young's modulus of 107 GPa and Poisson's ratio of 0.33 was used for the simulation of these components.[11]

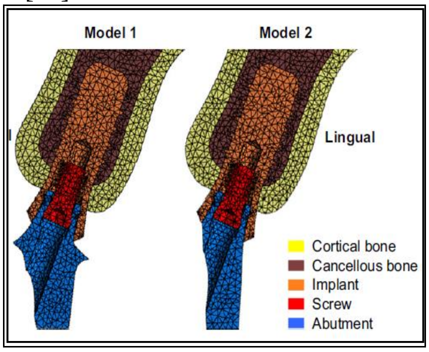

Fig. 4: Finite element models of Model 1(custom angled abutment design) and Model 2(conventional angled abutment design).

Material	Structure	Young's Modulus (GPa)	Poissons Ratio
Titanium	Implant	110.0	0.35
Titanium Alloy	Abutment	107.2	0.33

Table 1: Mechanical Properties

VI. 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. 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. Redesign co-existing design of implant abutment with washer.

VII. CONCLUSION

Different implant–abutment connection designs exhibited significant differences in survival time under dynamic loading and in maximum fracture strength. 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. CAD/CAM-produced implant frameworks are made from a solid block of material. With this specific fabrication technique, the material is more homogeneous and has high mechanical properties.

REFERENCES

[1] Sang-Choon Cho, Paula-Naomi Small, Nicolas Elian, and Dennis Tarnow "Screw Loosening for Standard and Wide Diameter Implants in Partially Edentulous Cases: 3- to 7-Year Longitudinal Data" Implant Dentist /Vol.3 Number 3 2004.

[2] Lars Steinebrunner, Stefan Wolfart, Klaus Ludwig, Matthias Kern "Implant–abutment interface design affects fatigue and fracture strength of implants" Clin. Oral Impl. Res. 19, 2008/1276–1284.

[3] Hussam El-Din El-Sheikh "Design Developeent of an Endosseous Dental Implant " J Prosthet Dent, 92:523-530, 2004.

[4] SR Desai, MS Desai1, G Katti2, I Karthikeyan "Evaluation of design parameters of eight dental implant designs: A two- dimensional finite element analysis''Nigerian Journal of Clinical Practice • Apr-Jun 2012 • Vol 15 • Issue 2 pg; 176-181.

[5] Theodoros Kapos, German O. Gallucci, Hans-Peter Weber "Computer Aided Design And Computer Assisted Manufacturing in Prosthetic Implant Destistry" The International Journal of Oral & Maxillofacial Implants Volume 24, Supplement, 2009.

[6] 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.

[7] 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

[8] 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.

[9] Marchack, C.B. "A custom titanium abutment for the anterior single-tooth implant" The Journal of Prosthetic Dentistry 76, 1996 , 288–291.

[10] Lars Steinebrunner , Stefan Wolfart, Klaus Ludwig "Implant–abutment interface design affects fatigue and fracture strength of implants" -Clin. Oral Impl. Res. 19, 2008; 1276- 1284.

[11] TingWua,WenheLiao,NingDai,Chunbo Tang "Design of a custom angled abutment for dental implants using computer-aided design and nonlinear finite element analysis"–Journal of Biomechanics 43 (2010) 1941–1946.

[12] Angel Alvarez –Arenal, Luis Segura-Mori, Ignacio Gonzal "Stress Distribution in the abutment Retention Screw of a Single Implant Supporing a Prosthesis with Platform Switching"-The International Journal of Oral & Maxillofacial Implants Volume 28, Number 3, 2013.

