ejpmr, 2018,5(1), 422-424

SJIF Impact Factor 4.161

EUROPEAN JOURNAL OF PHARMACEUTICAL AND MEDICAL RESEARCH

www.ejpmr.com

Review Article

ISSN 2394-3211

EJPMR

AN OVERVIEW OF FINITE ELEMENT ANALYSIS IN IMPLANT DENTISTRY

Sabzar Abdullah*[1], Geeta Rajput[2], Afzal Ahmad[3], Pranshu Varshney[4] and Mahbooba Khazir[5]

[1]Assistant Professor, Department of Prosthodontics, Dr. Z.A. Dental College, AMU, Aligarh.
[2]Professor, Department of Prosthodontics, Dr. Z.A. Dental College, AMU, Aligarh.
[3,4]Resident, Dr. Z.A. Dental College, AMU, Aligarh.
[5]Resident, Teerthanker Mahaveer Dental College and Research Centre, Moradabad.

***Corresponding Author: Dr. Sabzar Abdullah**
Assistant Professor, Department of Prosthodontics, Dr. Z.A. Dental College, AMU, Aligarh.

Article Received on 21/11/2017 **Article Revised on 12/12/2017** **Article Accepted on 01/01/2018**

ABSTRACT
Finite Element Analysis (FEA) is a modern tool for numerical stress analysis, with an advantage of being applicable to solids of irregular geometry that contain heterogeneous material properties. The history of Finite Element Analysis (FEA) dates back to 1943 when R. Courant first developed this technique. Finite Element Analysis (FEA) by structural analysis allows the determination of stress resulting from external force, pressure, thermal change, and other factors. This article provides a review of the achievements and extrapolating the concepts of Finite Element Analysis in Implant Dentistry.

KEYWORDS: Finite element analysis, finite element method, Geometric model, Three Dimensional, Implants.

INTRODUCTION
Stress analysis of dental structures has been a topic of interest in recent years with an objective of determining stresses in the dental structures and improvement of the mechanical strength of these structures. As we all know, oral cavity is a complex biomechanical system with limited access. Ever since its emergence and its subsequent application in the field of implant dentistry for the first time by Weinstein at el[1] in 1976, Finite Element Analysis (FEA) has emerged as a potent tool in the field of implant dentistry on account of its unprecedented ability to successfully simulate and predict the complexities and intricacies of the biomechanical behavior associated with stomatognathic system.[2]

For problems involving complicated geometries, it is very difficult to achieve an analytical solution. Therefore, the use of numerical methods such as FEA is required. Finite Element Analysis (FEA) is a technique for obtaining a solution to complex mechanical problem by dividing the problem domain into a much smaller and simpler domains (elements) in which the field variables can be extrapolated with the help of shape functions. In other words, Finite Element Analysis (FEA) is a method whereby, instead of seeking a solution for the entire domain, one formulates the solution functions for each finite element and combines them properly to obtain the solution to the whole body. Because the components in a dental implant-bone system are extremely complex geometrically, Finite Element Analysis (FEA) has been viewed as the most suitable tool for analyzing them.[3]

Factors responsible for the failure of any prosthesis can be biological as well as mechanical. Nevertheless, when it comes to dental implants the incidence of mechanical failure far outweigh the biological failure. For example, the development of a direct bone implant interface is largely biological. Most recent reports indicate the surgical phase of implants form a successful interface more than 95 percent of the time, regardless of the implant system used.[4] Hence, the biological aspect of the field is very predictable.

The most common implant complications are the biomechanical problems that occur after the implant is loaded. In recent years an increasing amount of materials used for dental implants are being fabricated using titanium and titanium alloys. The young's modulus of titanium is 5-10 times greater than that of cortical bone surrounding implants.[5] The fundamental engineering principle, composite beam analysis, express the concept that when two materials of different young's modulus are placed in direct contact with no intervening material and one material is loaded, a stress contour will be described at the point where the 2 materials come in contact. Thus, a stress contour develops in the peri-implant area surrounding the dental implant upon application of bite force during mastication.[6]

Physical Properties
Bite forces may have compressive, tensile or shear components. Compressive forces tend to push materials toward each other. Tensile forces pull objects apart. Shear forces on implant cause sliding. Bone is strongest

under compressive forces, 30% weaker under tensile loads and 65% weaker to shear forces.[7] Therefore, the most detrimental forces that can increase the stress around the bone implant interface are tensile and shear forces. These forces tend to harm material integrity and cause stress build up. During masticatory process, the repeated pattern of cyclic process transmits loading via the restoration and dental implants to the peri-implant bone. This generates different amount of stress around the ridge and also in the prosthetic structure.[8]

Implant Related Complications

The common causes of implant related complications are centered around stress. Thus overall treatment plan should be to (1) assess the greatest force factors in the system and (2) establish mechanism to protect the overall implant bone prosthetic system. The use of Finite Element Analysis (FEA) is gaining popularity because of its ability to accurately assess the complex biomechanical behavior of irregular prosthetic structures and heterogeneous material in a non-invasive, repeatable manner.

In implant dentistry literature, commonly used materials in Finite Element Analysis (FEA) studies can be classified as either implant, peri-implant bone(cortical and cancellous bone) and restoration. Finite Element Analysis (FEA) method allows application of simulated forces at specific points in the system and stress analysis in the peri-implant region and surrounding structures.[9]

The principal difficulty in simulating the mechanical behavior of dental implants is the modelling of human bone tissue and its response to applied mechanical force. Certain assumptions need to be made to make the modelling and solving process possible. The complexity of the mechanical characterization of bone and its interaction with implant system has forced authors to make major simplifications. Some assumptions influence the accuracy of Finite Element Analysis (FEA) results significantly.[10] These include (1) detailed geometry of bone and implant to be module (2) material properties(3) boundary conditions(4) the interface between zone and implant.[11]

The first step in FEA modelling is to represent the geometry of interest in computer in terms of points, lines, areas and volume. Now a days, this is accomplished by a 3D CT scanner for modelling of living tissues like bone and a 3D laser scanner for modelling inanimate objects.

The next step is conversion of geometric model to finite element model by discretization. Discretization is the process of dividing problem into several small elements, connected with nodes. All elements and nodes must be numbered so that a set-up of matrix connectivity is established. The elements could be one, two or three dimensional and in various shapes. It is essential that the elements are not overlapping but are connected only at the key points, which are termed nodes. The joining of elements at the nodes and eliminating duplicate nodes is termed as meshing. Several softwares options are currently available and can be used for Finite Element Analysis (FEA) mesh generation particularly Ansys (Swanson Analysis systems, Houston, PA, USA) and MSC/Nastram(MSC Software corporation, Santa Ana, CA, USA).[12]

Material properties which include those of living structures and mechanical non-living entities such as implant fixtures, abutments and restorations greatly influence the stress and strain distribution. These material properties must be incorporated in the FEA software model. The properties can be modeled in Finite Element Analysis (FEA) results as isotropic, transversely isotropic, orthotropic and anisotropic. In isotropic material, the properties are same in all directions; therefore only two material constants exist. An anisotropic material has different properties when measured in different directions. There are many material constants depending on the degree on anisotropy (transversely isotropic, orthotropic). In most reported Finite Element Analysis (FEA) based implant studies, the assumption is made that the materials are homogenous and linear and they have elastic material behavior characterized by two material constants of Young's modulus and Poisson's ratio. Thus, three factors namely Poisson's ratio, Young's modulus and density of the material when incorporated in the Finite Element Analysis (FEA) model will provide the software with data on how a given material behaves when submitted to force application taking into consideration its deformation capacity, elasticity and behavior under tension or compression.[13]

A boundary condition is the application of force and constraint. When a force is applied to a geometrical model constructed on the computer it will act like a free floating rigid body and will undergo a translation or rotatory motion or a combination of the two without experiencing deformation. To study its deformation some degrees of freedom must be restricted (movement of the node in each direction x, y and z) for some of the nodes called as Zero-displacement constraints, thus setting up the boundary conditions.

Most FEA models assume a state of optimal Osseointegration, meaning that the cortical and the trabecular bone are assumed to be perfectly bonded to the implant. This does not occur so exactly in clinical situations. Therefore, the imperfect contact and its effect on load transfer from implant to supporting bone need to be remodeled more carefully. Current Finite Element Analysis (FEA) programs provide several types of contact algorithms for simulation of contacts. It is therefore now technically feasible to conduct such a simulation. The friction between the contact surfaces can also be modeled with contact algorithms. The friction coefficients however have to be determined through experimentation.

Loading can be axial or non-axial. An axial force flows down the long axis of the implant and hence compresses the anchorage unit which is favorable. Non-axial or horizontal loading transmits tensile and shear stresses which tend to separate and bend the components which are considered destructive. For realistic stimulation of mastication, combinations of axial and non-axial forces are used.[12] Loading can also be static or dynamic. Dynamic loading although more realistic has been difficult to computationally model than static loading and hence most FEA use static loads which can be axial, non-axial or mixed. One study, comparing dynamic with static loading, revealed that dynamic loading resulted in greater stress levels than static loading. Dynamic loading has consistently been found to have more osteogenic potential than static loading.

Once force and time properties have been properly defined, the software performs a series of mathematical calculations and yield the results. These are presented as according to a color scale where each shade represents a different degree of movement, tension or compression. The model also allows selecting one particular axis or structure for the analysis of tension, compression or movement, allowing simulation of a variety of events and thereby increasing the possibilities of analysis.[13]

CONCLUSION

This article has attempted to address the basics of FEA in dental implantology from a practical view point. It is basically a numerical stress analysis that has been increasingly used now a days to evaluate the risk factors from biomechanical point of view. Nevertheless, simplification of problem on the basis of assumptions had stood as one of its limitation. So, there arises a need to validate its results in clinical settings. By understanding its basics, method of application as well as limitations, the researcher can effectively extrapolate its results to clinical situations.

REFERENCES

1. Weinstein AM, Klawitter JJ, Anand SC, Schuessler R. Stress analysis of porous rooted dental implants. J Dent Res., 1976; 55: 772-7.
2. Jian-Ping, Keson B.C. Tan, Gui-Rong Liu. Application of finite element analysis in implant dentistry: A review of literature. Journal of Prosthetic Dentistry, June 2001; 85(6): 585-598.
3. Carl E. Misch. Contemporary Implant Dentistry, Third edition; Mosby, Elsevier, page 68-88 & 544-546.
4. Goodacre CJ, Bernal G., Rungcharassaeng K et al Clinical Complications with implants and implant prosthesis. Journal of Prosthetic Dentistry, 2003; 90: 121-132.
5. Reilly DT, Burstein AH: The elastic and ultimate properties of compact bone tissue. J. Biomechanic, 1975; 8: 393-405.
6. Baumeister T, Avallone EA (1978) Marks' standard handbook of mechanical engineers, ed8, new york, Mc Graw Hill.
7. Thompson MC, Field CJ, Swain MV. The all ceramic, inlay supported fixed partial denture. Part 2. Fixed partial denture design: a finite element analysis. Australian Dental Journal, 2011; 56: 301-11.
8. B.Alper Gultekin, Pinar Gultekin and Serdar Yalcin. 2012 Gultekin et al., licensee In Tech Chapter 2, Application of Finite Element Analysis in Implant Dentistry.
9. Korioth TW, Versluis A.Modelling the mechanical behavior of the jaws and their related structures by finite element(FE) analysis Crit Rev Oral Biol Med., 1997; 8: 90-104.
10. Van Oosterwyck H, Duyck J, Vander sloten J, Van der Perre G, De Cooman M, Lievens S et al. The influence of bone mechanical properties and implant fixation upon bone loading around oral implants. Clin Oral Implants Res., 1998; 9: 407-18.
11. Prasad Konda & Tarannum SA.Basic principles of Finite Element Method and its application in Orthodontics. Journal of pharmaceuticals and Biomedical Sciences, 2012; 16(11).
12. Zhan JK, Chen ZQ(1998)The study of the effects of changes of the elastic modulus of the materials substitute to human hard tissues on the mechanical state in implant bone interface by three dimensional anisotropic finite element analysis. West China J Stomatol, 1998; 16: 274-8.
13. Akuz E, Braun TJ, Brown NA922060Static versus dynamic loading in the mechanical modulation of vertebral growth. Spine, 31: 952-958.