ISSN No:-2456-2165

Finite Element Analysis and Application in Dental Implantology

Ahmet Aktı
Department of Oral and Maxillofacial Surgery
Selcuk University Faculty of Dentistry
Konya/Turkey

Doğan Ilgaz Kaya
Konya Oral and Dental Health Hospital
Konya/Turkey

Abstract:- It is not always possible to carry out many studies planned in all medical fields including dentistry due to ethical reasons, high costs and technical impossibilities. Finite elements methods (FEM), which has been used for many years in the field of engineering, has prevented such impossibilities for more than 40 years in dentistry and medicine, and enables complex researches that are quite difficult to perform under normal conditions, in a computer environment. Finite elements methods can bring fast and easy solutions to many problems in the field of health, especially in dental implantology, including strength, stress and material science, including engineering. Finite elements methods provides fast and reliable results on the subjects studied, but it is an important guide for many clinical studies. In this review, it is aimed to obtain general information about the finite element stress analysis method and to examine the studies on its usability and reliability in the field of dental implantology.

Keywords:- *Finite Element Method, Stres Analysis, Dental Implantology.*

I. INTRODUCTION

In order to evaluate the biomechanical properties of the materials used for therapeutic purposes in dentistry and to determine the stresses formed in the dental structures and adjacent tissues, stress analysis of these structures has become quite common in recent years. Knowing the mechanical properties and stresses of the materials used in dentistry is very important for a successful treatment[1]. It can be difficult, expensive, risky and sometimes impossible to determine how living tissues and organs behave against forces, and to determine stress analysis [2]. It is not possible to carry out the planned studies in dentistry due to high cost, ethical and technical difficulties. For this reason, stress analysis studies may need to be performed on models that represent living tissues. Various stress analysis methods are used to predict what the design or combination should be in order to see the regions where the forces on an object are concentrated and to make it stronger and more durable against the forces that can come [3]. These stress analysis methods;

- Photoelastic stress analysis method
- Tension meter stress analysis method
- Stress analysis with brittle varnish coating method
- Force analysis with holographic interferometry (Laser Beam)
- Thermographic force analysis method
- Force analysis method with radiotelemetry
- Finite element stress analysis method

II. FINITE ELEMENT ANALYSIS TECHNIQUE

Finite element analysis is an analysis technique based on the basic mathematical model of the biomechanical system and analysis of this model by computer. The technique, which operates according to the principle of going from piece to whole, can also be expressed as an imitation of the natural state of nature on the computer. Finite element analysis is one of the most modern and important scientific techniques of today, bringing numerical solutions to mathematical equations describing physical models. It was first developed for the solution of structural problems in the aircraft and space industry, especially in the 1950s, and is now widely used in many different fields such as performing static analyzes, fluid mechanics, heat transfer analysis, conducting electromagnetic analysis [4,5,6]. It is widely used in medicine and dentistry today, as in most branches of engineering. The first known study about finite element analysis in dentistry is the research of Ledley and Huang in 1968. In this study, a tooth with a mathematical model was applied to forces in different directions and the stresses caused by these forces in the bone tissue surrounding the tooth were evaluated[7].

The structure to be analyzed by finite element analysis technique is divided into finite number elements and its response to force is examined numerically. One, two or three dimensional analysis of a structure analyzed by this technique can be performed. In the model created through computer programs, shape changes, stress distribution and intensities that occur depending on the applied force, direction and force application in the area are determined[5,8]. The problem, which is desired to be solved with the finite element analysis method, is divided into simple sub-units even if it is complex and each structure is solved within itself. Even if the problem desired with finite element analysis is complex, each structure is solved within itself by dividing it into simple subunits called finite elements. The problem, which is simplified with a large number of small and interconnected subunits, is solved by the principle of going from piece to piece [9].

ISSN No:-2456-2165

Advantages of finite element analysis[10,11];

- Complex solids can be modeled.
- Stress distributions can be evaluated in detail.
- Allows analysis of structures made up of different materials.
- Real definitions can be obtained by making physical definitions of materials.
- Craniofacial and dental structures can be simulated.
- It can be repeated.
- It results in a shorter time than experimental research.

Despite the numerous advantages of finite element analysis, it also has some disadvantages. These disadvantages [10,11,12,13];

- High cost of computers and software equipped with the analysis
- The necessity of equipped and experienced personnel who can use the software to be analyzed
- The obligation to update the software programs used with the developing technology
- The necessity of transferring the live system to the virtual environment in full detail for the accuracy of the research conducted
- The assumptions of the properties of the modeled material, such as isotropic, homogeneous and linear elasticity, do not fully reflect the reality.

III. STAGES OF FINITE ELEMENT ANALYSIS TECHNIQUE

Finite element analysis is carried out in 3 stages:

- *Pre-Processing:*

The first step is to create the geometric model of the structure for analysis. After the model is created, the area is divided into elements and a mesh model is created. The higher the number of staff, the closer to the real results can be obtained in the analysis. In an analysis process using the finite element analysis method, networking is the basis of the analysis. With the networking process, the coordinates of the elements and the nodes connecting these elements are created [14]. The points where the elements are connected to each other are called nodes, and the entire structure is called mesh.

While creating the network model, neighboring elements do not overlap and there is no space between them. The structure of the elements should be as simple as possible. Lines in one dimension, triangles or parallel edges in two dimensions; in three dimensions, four, five and six faced structures are preferred. One-dimensional objects are divided into finite elements by nodes, two-dimensional objects by lines, three-dimensional objects by planes. In all cases, the elements representing the object are connected by nodes. As a result, the body will be replaced by a system of finite elements and nodes connecting them. The term “body” in general; the structure is used to mean the continuous environment or the region of the problem. Nodes, on the other hand, can be compared to nut and bolt connections that connect adjacent finite elements at their ends and hold them together. Since the nodes are removed, the physical continuity is lost between adjacent finite elements, since the elements will separate [14]. The next step in analyzing the method is to define the element matrices for each of the elements representing the object. The element matrices are then summed to form the general matrix of the entire segmented object. In this collection, the balance of forces and continuity of displacements are provided in all nodes in the finite element model of the body [14]. Then the boundary conditions are applied to the model. Boundary conditions include boundary expressions of stresses and displacements. It shows where the object is fixed and where the force is applied. It is determined by the state of the object. Boundary conditions are determined according to which region of the analyzed object is applied [5].

- *Solution:*

It is the stage where the equations formed between the finite number of elements after the forces are applied are solved and resolved [15].

- *Evaluation of Results Stage (Post-Processing):*

It is the stage where the solution of the equations is visualized by means of charts, figures and color graphics [16].

IV. BASIC MECHANICAL CONCEPTS USED IN FINITE ELEMENT ANALYSIS

A. Force

The effect that can change the motion and shape of objects is called force. The effect made by other objects on the object under study can be defined as external force, and the effect and reaction force between the various parts of the object can be defined as internal force. Force is a scalar magnitude and has vectorial properties such as direction, direction and violence. There are 3 types of forces: compression, stress and shear-type forces.

B. Stress

Tension is the reaction of an object to the applied force in the opposite unit area. With tension, greater effects occur in the internal structure of the body than the molecular structure. Tension in a structure means the force per unit surface. Its unit is pascals, kg / cm2 or N / m2 [17,18].

Stress = Strength / Area

There are 3 types of stress (stress) depending on the applied force:

1) Tensile stress: It is the tension that is perpendicular to the surface of the object and forces its molecules to be separated, created by two forces in the same direction and in the opposite direction. Tensile stress is the force that occurs against distortion caused by the force that wants to stretch or stretch objects.

ISSN No:-2456-2165

2) Compressive stress: When an object is subjected to a force that tries to squeeze or shorten itself, the stress caused by this force is called compression stress.
3) Shear stress: It is the type of tension formed by two forces at different levels and opposite directions, forcing the molecules of the body to slide on each other like the layers, parallel to the surface and in opposite direction. It is known that cutting (shear) stresses are the most destructive stresses for fixation materials.

C. Strain

The ratio of the dimensional change that occurs as a result of the force applied to an object to the original size of the object is called strain. It can also be expressed in the form of deformation occurring at any point of the body under loading. It shows how much change has been applied after the force has been applied compared to the situation before the force has been applied to the material. Strain is usually expressed in percent (%). If the applied force is larger than the tension that the object can withstand, the force that holds the building blocks together will exceed the force, which may cause rupture or breakage [19].

Strain = Shape change / Original length

D. Principal Stress

The stresses in which the existing stresses act only perpendicularly and the shear stress does not affect in any plane is called principal stress. Prime stresses; It is divided into three as maximum, intermediate and minimum principal stress. Maximum principal stresses (Pmax) refer to tensile stresses and are positive. Minimum principal stresses (Pmin) express the highest stress (compression) stresses and are negative [20,21].

E. Von-Mises Stress

Von Mises stress is used for retractable materials and is defined as the beginning of deformation. It is used to obtain information about stress distributions and concentrations on the material. By combining stresses that occur in two or three dimensions, it gives the tensile strength of the material loaded in one direction. Von Mises is also used in the analysis of measuring stress and fracture resistance [22].

F. Elasticity Module (Young Module)

It is a measure of the resistance of an object to deformation. As the rigidity of the objects increases, the modulus of elasticity rises. Its unit is Pascal (N / cm2). It is the strain ratio of stress applied to any point of the object. The elasticity module or elastic coefficient is also called the "Young module" since it was first calculated by the English physicist Thomas Young [23,24].

G. Poisson Ratio

When the pulling force is applied, the length of the section decreases while the length of the object is applied, while the length of the object is applied, the section thickness increases. Under tensile and compressive forces, the ratio of the lateral dimension change of the body to the axial dimension change is called the poisson ratio (v). Poisson ratio varies depending on the material. This rate does not have a unit. The reduction in cross section continues until the material breaks. Softer materials show more reduction in cross-section during drawing, and the poisson ratio is higher [1].

H. Hook's Law

It is a law that accepts a linear relationship between voltages and strains. It expresses the behavior of objects approximately, provided that they do not exceed certain stress limits. The curve, which shows the relationship between tension and stress, is used to predict how much distortion will occur when the force is applied to the object. The straight slope in this curve shows the stiffness of the object and gives the force coefficient (k). He says that rigid materials have a high elasticity coefficient, and flexible materials have a low elasticity coefficient. Its formula is (F = -kx). The negative sign in the formula indicates that the force is always in the direction of displacement [25].

I. Deformation

Temporary (elastic deformation) or permanent (plastic deformation) deformation occurs after an object is applied. As a result of tensile forces, deformation occurs in the form of elongation, shortening and compression occurs in the compressive forces [20].

J. Fatigue Resistance (Yield Stress)

The material whose maximum resistance is exceeded is broken, that is, it is subjected to plastic deformation. The limit between elastic and plastic deformation is called fatigue resistance. Up to the point of Yield, the object is in elastic deformation and can be restored if the stress disappears [19].

K. Flexion

It is called moving around an axis as a result of the force applied to the object. Mutual compression and tension forces occur on the bent surfaces of the object [19].

L. Isotropic Material

When the force is applied from different directions, they are the same materials that show the same mechanical properties and have the same elastic properties in all directions regardless of their coordinate system [19].

M. Orthotropic Material

They are substances that show different mechanical properties when force is applied from different directions. Its elastic modules vary depending on the direction in which the force is applied. Bone tissue can be given as an example [19].

N. Element (Element)

Simple geometric shapes used in finite element analysis are called elements. Workers; According to their geometry and dimensions such as triangle, parallel edge, and rectangle, they can be classified as one-dimensional, two-dimensional, rotating elements, three-dimensional elements [26].

O. Node

In finite element analysis, the points where elements whose behavior has been determined on the model made in computer environment are called nodes [27].

V. RESEARCHES IN DENTAL IMPLANTOLOGY USING FINITE ELEMENT ANALYSIS TECHNIQUE

There are many studies in the field of dental implantology conducted in the literature with the finite element analysis technique. In their study, Melo et al [28] compared the 2.9 mm narrow implants to be placed in the back of the all-four implant system to support the fixed prosthesis in the maxilla in terms of stress with 3.5 mm implants. In their study, Bhering et al [29] compared all-on-four and all-on-six treatment concepts for the prosthetic rehabilitation of atrophic maxilla. In their study, Wen et al [30] compared 3 different zygomatic implant techniques in serious atrophic maxilla. In their study, Eom et al [31] compared the implant-supported prosthetic treatment option and different prosthetic treatment options in terms of stress. De Souza et al. [32] compared the cantilever extension and other treatment options in implant-supported prostheses in the posterior maxilla. In their study, Silva et al. [33] compared complete toothless rehabilitation with 6 implants with all-on-four technique in the toothless maxilla with the finite element analysis technique and reported that more stress occurred in the all-on-four technique, especially if there was a quantilever, more stress accumulated in the quantilever parts. In a study conducted by Baiamonte et al. [34] to test the reliability of finite element analysis studies in terms of dental implantology, the strain values occurring in the monkey mandible with an osseointegrated titanium implant were evaluated by both in vitro and finite element analysis studies. Based on the high overlap of the results obtained, the researchers concluded that the finite element stress analysis method can be safely applied in dental implantology. In their study, Akça et al. [35] compared the finite element analysis and the stran gauge analysis method to examine the stresses on dental implants. In both methods, close results were obtained about the amount of stress occurring under the applied forces, but in terms of modeling, the finite element analysis method was found to be more advantageous because it allows more sensitive and detailed results.

VI. CONCLUSION

As in many fields, the finite element stress analysis method, which has been used for many years in medicine, is also widely used in the field of dental implantology. It is a useful method especially in terms of enabling unlimited repetition of researches that cannot be repeated clinically in one-to-one conditions with different scenarios and enabling researches that cannot be applied on patients for ethical reasons without ethical responsibility. It enables many new techniques to be researched and developed without being applied on the patient. In addition, in many studies, similar results with in vitro methods increase confidence in the technique. However, it should not be forgotten that according to today's technology, it is not possible to dynamically transfer all the details of natural conditions to the computer model. For this reason, it is beneficial to confirm the finite element analysis studies on living tissues with clinical studies.

REFERENCES

[1]. Craig R, Powers J, 2002. Restorative Dental Materials 11th. Mosby St Louis

[2]. Magne P, 2007. Efficient 3D finite element analysis of dental restorative procedures using micro-CT data. Dental materials, 23, 5, 539-48.

[3]. Çalıkoğlu S, 1992. Departmental Prosthetics, 2nd Edition, Istanbul Ü Printing House, Istanbul.

[4]. Holmgren EP, Seckinger RJ, Kilgren LM, Mante F, 1998. 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. Journal of Oral Implantology, 24, 2, 80-8.

[5]. Geng J-P, Tan KB, Liu G-R, 2001. Application of finite element analysis in implant dentistry: a review of the literature. The Journal of prosthetic dentistry, 85, 6, 585-98.

[6]. Ebrahimi F, 2012. Finite Element Analysis: New Trends and Developments, BoD – Books on Demand.

[7]. Ledley RS, Huang HK, 1968. Linear model of tooth displacement by applied forces. Journal of dental research, 47, 3, 427-32.

[8]. Sonugelen M, Artunç C, 2002. Oral Prosthetics And Biomechanics, Ege University.

[9]. Mackerle J, 2004. Finite element modeling and simulations in dentistry: a bibliography 1990–2003.Computer methods in biomechanics and biomedical engineering, 7, 5, 277-303.

[10]. Farah J, Craig R, Meroueh K, 1988. Finite element analysis of a mandibular model. Journal of oral rehabilitation, 15, 6, 615-24.

[11]. Daegling DJ, Hylander WL, 2000. Experimental observation, theoretical models, and biomechanical inference in the study of mandibular form. American Journal of Physical Anthropology: The Official Publication of the American Association of Physical Anthropologists, 112, 4, 541-51.

[12]. Oehlers DJ, Seracino R, Yeo MF, 2000. Effect of friction on shear connection in composite bridge beams. Journal of Bridge Engineering, 5, 2, 91-8.

[13]. Pepper DW, Heinrich JC, 2005. The finite element method: basic concepts and applications, Taylor & Francis.

[14]. De Vree J, Peters M, Plasschaert A, 1983. A comparison of photoelastic and finite element stress analysis in restored tooth structures. Journal of oral rehabilitation, 10, 6, 505-17.

[15]. Hughes TJ, (1987). Hughes, The finite element method. Linear static and dynamic finite element analysis. With the collaboration of Robert M. Ferencz and Arthur M. Raefsky, Prentice Hall.

[16]. Siegele D, Soltesz U, 1989. Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. International Journal of Oral & Maxillofacial Implants, 4, 4.
[17]. Maurer P, Holweg S, Schubert J, 1999. Finite-element-analysis of different screw-diameters in the sagittal split osteotomy of the mandible. Journal of cranio-maxillofacial surgery, 27, 6, 365-72.
[18]. Jafari A, Shetty KS, Kumar M, 2003. Study of stress distribution and displacement of various craniofacial structures following application of transverse orthopedic forces — a three-dimensional FEM study. The Angle Orthodontist, 73, 1, 12-20.
[19]. Fung Y-c, 2013. Biomechanics: mechanical properties of living tissues, Springer Science & Business Media.
[20]. Marghitu DB, 2001. Mechanical engineer's handbook, Elsevier.
[21]. Sung SJ, Baik HS, Moon YS, Yu HS, Cho YS, 2003. A comparative evaluation of different compensating curves in the lingual and labial techniques using 3D FEM. American journal of orthodontics and dentofacial orthopedics, 123, 4, 441-50.
[22]. Incropera FP, Dewıtt DP, 2002. Fundamentals of Heat and Mass Transfer, New York, John Wiley, p. 93-142.
[23]. Yalçın H, Gürü M, 2002. Material information, Palme Publishing Distribution.
[24]. Misch CE, 2014. An implant is not a tooth: a comparison of periodontal indices. Dental Implant Prosthetics-E-Book, 46.
[25]. Rho JY, Ashman RB, Turner CH, 1993. Young's modulus of trabecular and cortical bone material: ultrasonic and microtensile measurements. Journal of biomechanics, 26, 2, 111-9.
[26]. Moaveni S, 2011. Finite element analysis theory and application with ANSYS, 3 / e, Pearson Education India.
[27]. Kutz M, 1998. Mechanical engineers' handbook, Wiley.
[28]. de Melo Jr EJM, Francischone CE, 2019. Three-dimensional finite element analysis of two angled narrow-diameter implant designs for an all-on-4 prosthesis. The Journal of Prosthetic Dentistry.
[29]. Bhering CLB, Mesquita MF, Kemmoku DT, Noritomi PY, Consani RLX, Barão VAR, 2016. Comparison between all-on-four and all-on-six treatment concepts and framework material on stress distribution in atrophic maxilla: A prototyping guided 3D-FEA study. Materials Science and Engineering: C, 69, 715-25.
[30]. Wen H, Guo W, Liang R, Xiang L, Long G, Wang T, Deng M, Tian W, 2014. Finite element analysis of three zygomatic implant techniques for the severely atrophic edentulous maxilla. The Journal of prosthetic dentistry, 111, 3, 203-15.
[31]. Eom J-W, Lim Y-J, Kim M-J, Kwon H-B, 2017. Three-dimensional finite element analysis of implant-assisted removable partial dentures. The Journal of prosthetic dentistry, 117, 6, 735-42.
[32]. de Souza Batista VE, Verri FR, Almeida DAdF, Santiago Junior JF, Lemos CAA, Pellizzer EP, 2017. Finite element analysis of implant-supported prosthesis with pontic and cantilever in the posterior maxilla. Computer methods in biomechanics and biomedical engineering, 20, 6, 663-70.
[33]. Silva GC, Mendonca JA, Lopes LR, Landre Jr J, 2010. Stress patterns on implants in prostheses supported by four or six implants: a three-dimensional finite element analysis. International Journal of Oral & Maxillofacial Implants, 25, 2.
[34]. Baiamonte T, Abbate M, Pizzarello F, Lozada J, James R, 1996. The experimental verification of the efficacy of finite element modeling to dental implant systems. The Journal of oral implantology, 22, 2, 104-10.
[35]. Akça K, Çehreli MC, İplikçioğlu H, 2002. A comparison of three-dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. International Journal of Prosthodontics, 15, 2.