ISSN 2616-5791 Vol.4, Issue 10: 66-72, DOI: 10.25236/AJMHS.2023.041011

Research progress in the application of three-dimensional finite element method in the field of dentistry

Zhang Yulu[1], Liu Jia[2,*]

[1]Department of Children's Stomatology, First Affiliated Hospital of Xinjiang Medical University (Affiliated Stomatology Hospital), Urumqi, Xinjiang Uygur Autonomous Region, 830054, China
[2]Xinjiang Uygur Autonomous Region Institute of Stomatology, Urumqi, Xinjiang Uygur Autonomous Region, 830054, China
[*]Corresponding author: liujia_0806@163.com

Abstract: *The three-dimensional finite element method is a theoretical mechanical application analysis method related to modern computer technology, and is also an important means in biomechanical research. It has a wide range of applications in the field of dentistry. Therefore, the application of the three-dimensional finite element method to solve biomechanical problems in dentistry has received increasing attention. This article provides a review of the current research status of three-dimensional finite element method in the fields of oral medicine, oral restoration, implantation, orthodontics, and oral and maxillofacial surgery in recent years.*

Keywords: *three-dimensional finite element method; Stomatology; Biomechanics; Stress analysis*

Finite element method (FEM) is a numerical method for analyzing structural stress and deformation. Its basic idea is to divide a complex geometric body into smaller and simpler finite elements, which are analyzed and then integrated to obtain the solution of the entire complex geometric body. FEM has become an important tool for analyzing and predicting the stress and strain distribution of natural teeth, dentures, implants, and surrounding bones. Each part of the research object's geometric characteristics, material properties, boundary conditions, loads, interfaces, and convergence can be calculated. By simulating areas in the body that are difficult to measure or touch, FEM can predict the actual clinical situation without creating real clinical samples and conditions, significantly reducing costs[1]. FEM experiments have high repeatability, and their application is particularly important under certain ethical constraints and experimental conditions that are difficult to achieve[2]. In recent years, the advancement of computer technology has led to rapid development in related research. FEM modeling has evolved from 2D to 3D, material properties have been set from linear to nonlinear, and modeling methods have shifted from simple computer modeling to the introduction of computer tomography (CT) or magnetic resonance imaging (MRI) to establish finite element mesh models. Simulation objects have shifted from only simulating hard tissues to being able to reconstruct surrounding soft tissues and periodontal membranes. This article reviews the research progress of three-dimensional finite element analysis in the field of dentistry in recent years.

1. Application of three-dimensional finite element analysis in the field of dental pulp disease

3D finite element analysis can establish a geometric model that reproduces solid teeth, verify whether some research conclusions are scientific and practical, and provide a good experimental basis for biomechanical research on the treatment of dental pulp diseases. In root canal therapy, pulp opening is an important prerequisite for achieving the expected goals of root canal therapy. Maximizing the preservation of remaining tooth tissue and achieving the integrity of the remaining tooth structure is of great significance for long-term tooth retention after root canal therapy[3]. The three-dimensional finite element analysis method was used to analyze the maximum principal stress, von Mises stress, and improved von Mises stress distribution in the finite element models of the traditional open pulp resin filling group, the traditional post open pulp all ceramic crown repair group, the minimally invasive open pulp resin filling group, and the minimally invasive post open pulp all ceramic crown repair group. The results suggest that minimally invasive pulp opening should be used as much as possible for the upper central incisor pulp opening, and crown repair is recommended after traditional pulp opening, Minimally

ISSN 2616-5791 Vol.4, Issue 10: 66-72, DOI: 10.25236/AJMHS.2023.041011

invasive posterior coronal repair has no significant advantage[4].

By simulating an immature premolar model through three-dimensional finite element analysis, the filling effect and apical sealing effect of calcium hydroxide, mineral trioxide aggregate (MTA), and biodentine were compared. The results indicate that compared with MTA and Biodentine dentin biological repair cement, the filling effect of calcium hydroxide is not ideal; MTA or Biodyne increases the stress at the root tip and reduces the stress inside the root canal; The use of a combination of calcium hydroxide, MTA, and Biodentine composite resin for repair can achieve the effect of preserving both the coronal and root structures simultaneously[5]. Some scholars have established a mandibular molar model to compare the stress distribution of MTA pulp capping with different thicknesses when repairing different sizes of perforations. The results show that the stress and strain at the MTA pulp interface are inversely proportional to the thickness of MTA, indicating that thicker MTA pulp capping can be used in clinical practice to reduce the stress at the interface between the repair material and pulp[6].

Kim et al.[7] explored the presence or absence of inverted filling material and the effect of the size of the prepared apical area on stress by constructing a three-dimensional finite element model of the mesial root of mandibular molars. The results showed that stress decreases with the increase of the preparation area when there is inverted filling material, and increases with the increase of the preparation area when there is no inverted filling material. Clinically, it is suggested that doctors should maintain the balance of dentin on both sides of the apical preparation area to reduce stress. Avoid rinsing or removing the inverted filling material, which may lead to surgical failure. Ran et al.[8] established a finite element model to investigate the stress distribution and tooth movement under different occlusal states after apical resection of maxillary central incisors. The study found that when the apical resection was more than 6 mm, the stress and strain in the apical region significantly increased, and it would also exacerbate the problems of deep overbite and deep overbite; In the case of deep coverage, the stress and tooth displacement significantly increase when the apical resection exceeds 5mm; In the case of deep overbite, the stress and tooth displacement significantly increase when the apical resection exceeds 4 mm, indicating that clinical doctors need to comprehensively consider the impact of resection length when performing apical resection.

2. Application of three-dimensional finite element analysis in the field of periodontal disease

The periodontal ligament is the soft tissue that connects teeth and alveolar bone, playing a crucial role in the physiological movement of teeth and being the most important factor in the inflammatory damage of periodontal tissue. An accurate finite element model can be used to establish a periodontal ligament model using micro-CT. Most studies have explored the distribution of tooth mobility and stress in periodontal tissue under physiological chewing and traumatic loading conditions.

Zhang et al.[9] studied the establishment of a finite element model of the mandibular first molar using CT images, and analyzed the influence of the area, position, and direction of the occlusal load area on the stress of the tooth and periodontium. The results showed that the stress patterns of the teeth and periodontium were significantly different. During the same chewing process, the stress on the tooth is significantly higher than that on the periodontal tissue, and the area of the stress concentration area on the tooth gradually decreases from the outside to the inside. This suggests that a reasonable design of occlusal contact can help improve the stress distribution on the tooth and periodontal tissue, thereby reducing load-related structural problems.

Most studies have established the periodontal ligament as an extremely thin tissue surrounding the tooth root, but simplified models cannot truly reflect the physiological movement of teeth. Tuna et al.[10] improved the periodontal ligament model by utilizing a contact model to reduce modeling time, increase accuracy, and improve the smoothness of the model. At the same time, the periodontal ligament is a porous fibrous soft tissue located around the teeth, playing a crucial role in transmitting loads from the teeth to the alveolar bone of the mandible. Ort ú n Terrazas et al.[11] established a material model similar to periodontal ligament, which has porous and fiber structure characteristics similar to periodontal ligament. Through rigorous tensile and compressive load tests, it was found that the bulk modulus of porous materials resembling periodontal membranes is related to the drainage capacity of pore fluids, and porous materials play an important role in the drainage of tissue fluids during the compression process;During the stretching process, collagen fibers moving along the load direction are an important factor affecting the stiffness of the periodontal ligament, indicating that the integrity of the periodontal ligament structure is the main factor in maintaining periodontal health.

ISSN 2616-5791 Vol.4, Issue 10: 66-72, DOI: 10.25236/AJMHS.2023.041011

3. Application of three-dimensional finite element analysis in the field of orthodontics

Orthodontics focuses on studying the stress distribution and patterns caused by the transmission of orthodontic forces, and exploring the mechanisms of orthodontic treatment.In orthodontic cases of malocclusion, the use of micro braces has good therapeutic effects, but its high failure rate is the main problem. The angle of planting, the type of micro anchorage, and the direction of stress have a significant impact on the stress area and magnitude[12]. Improper design and uneven force distribution can directly generate stress on the screw, affect the stability of the anchorage, stimulate surrounding tissues, and cause inflammation. A three-dimensional finite element study has shown that the new type of micro anchorage can effectively reduce the stress of fixed screws[13]. Lee et al.[14] found that bicortical microbranches can reduce the deformation and fracture of the anchorage, achieve more expansion in the coronal plane, and significantly improve stability.

Liu Qinghui et al.[15] established a three-dimensional finite element model of the maxillary alveolar bone periodontal membrane maxillary central incisor invisible appliance. Through FEM, it was found that when the maxillary central incisor lingual movement occurs, the periodontal membrane of the central incisor without attachments is subjected to greater force than that of the group with attachments, and the periodontal membrane of the attachment body with optimized bonding (upper half elliptical attachment, upper half tetrahedral attachment) is subjected to less force than that of the ordinary attachment body (horizontal rectangular attachment, horizontal elliptical attachment). The orthodontic treatment of impacted canine teeth on the palatal side requires strict anchorage control and a long treatment time.Therefore, accurate positioning and application of force are the key to successful treatment. Scholars have established finite element models to study the stress distribution of impacted canine teeth under different degrees of initial force and in different directions (buccal, vertical, and distal). The results show that the stress generated by applying vertical force is the lowest, and the stress distribution on the root varies with different initial tilt forces. At the same time, it indicates that the stress generated by buccal force in the neck of the tooth will resist movement, and applying vertical force and distal tilt force is more conducive to tooth movement than applying buccal force, especially for severely tilted canines, the difference is more significant[16]. Nagendraprased et al.[17] studied the displacement patterns and periodontal membrane stress of impacted canines and their adjacent lateral incisors and first premolars under applied force. The results showed that there were differences in displacement and stress distribution of impacted canines at different angles. Under the same stress, when the impacted canines on the palatal side tilt towards the mesial axis, the displacement of the maxillary canines decreases. As the force increases, the stress on the periodontal membrane also increases, indicating the use of minimum force to pull impacted canines in clinical orthodontic or other treatments. Moga et al.[18] found that rotational orthodontic force causes the greatest pressure, while translational force produces the smallest pressure on capillaries. If the area of the periodontal ligament decreases, the periodontal ligament has a stronger ability to bear stress under smaller orthodontic forces.

4. Research on the Application of 3D Finite Element Analysis in the Field of Dental Prosthetics

Oral restoration aims to restore the missing or missing dentition and its physiological function in patients with missing teeth. Designing a targeted and reasonable restoration plan can minimize iatrogenic tooth fractures and significantly improve the service life of the affected teeth after restoration. Dejak et al.[19] found that the force on the anterior teeth with palatal/lingual inclination was the smallest, and the greater the labial inclination angle of the anterior teeth, the greater the force. The force on the labial inclination anterior teeth repaired with post core significantly increased compared to the labial inclination anterior teeth at the same position. By constructing a three-dimensional finite element model and analyzing the effect of tip coverage thickness on the stress distribution of premolars restored with all ceramic high inlays, it was found that increasing the tip coverage thickness of all ceramic high inlays can reduce the risk of all ceramic high inlay fracture, but may lead to high inlay detachment and palatal dentin fracture[20]. Zhang et al.[21] analyzed the influence of different edge thicknesses, degrees of polymerization, and adhesive thicknesses on the fracture strength of glass ceramic materials through a three-dimensional finite element model. The study found that the thicker the edge of the restoration, the smaller the degree of polymerization, the greater the resistance of glass ceramic to fracture. This confirmed that the thickness of the restoration is directly proportional to the fracture resistance, and the polymerization angle is inversely proportional to the fracture resistance.

Miura et al.[22] simulated the effects of four bracket designs on the mechanical performance of a three unit cantilever fixed denture, and compared the conventional design group, the bracket design group with

ISSN 2616-5791 Vol.4, Issue 10: 66-72, DOI: 10.25236/AJMHS.2023.041011

a 2mm extension towards the buccal lingual side, the bracket design group with a 0.5mm increase from the base of the abutment to the connecting area, and the combination design group with the latter two groups. The results showed that the combined design group had a more balanced force, avoided stress concentration, and had a protective effect on the abutment. Shahmiri et al.[23] found through three-dimensional finite element analysis that the I-type rod design has stronger support, while the bracket design of the distal occlusal support has stronger support for the removable denture base compared to the proximal occlusal support. Chen et al.[24] conducted biomechanical analysis on the restoration methods of tooth support and implant natural tooth joint support, and found that the sleeve crown design with two implants implanted at the free end and combined with natural teeth supports the abutment teeth more evenly. This suggests that in clinical practice, if the free end is missing, at least two implants should be implanted at the free end as much as possible to extend the service life of the restoration.

5. Research on the application of three-dimensional finite element analysis in the field of planting science

The biomechanical properties that implant dentures should possess when exercising chewing function, as well as the ability of the implant to firmly bond with the jawbone, are crucial for the success of implantation. The application of three-dimensional finite element analysis in dental implant science is one of the current research hotspots.The application of three-dimensional finite element analysis can study the stress patterns of various components of the implant and the bone tissue around the implant. Research has shown that the number, diameter, length, thread shape, material of implants, as well as the quality and quantity of the surrounding bone tissue themselves, affect the stress mode of the bone tissue around the implant[25]. In the mechanical research of implants, Von Mises stress, maximum/minimum principal stress, and maximum shear stress are often used for evaluation. Von Mises stress represents the stress distribution inside the model, the maximum/minimum principal stress can evaluate the tension and pressure situation of the model, and the maximum shear stress indicates that the model is in a critical state of failure under complex stress[26]. As is well known, mechanical stress has a significant impact on the homeostasis of bone tissue. Animal experiments have also shown that excessive occlusal stress can lead to loss of bone tissue around the implant[27].

In a long-term study on cases of dental implant supported overdentures,the incidence of non inflammatory bone loss was much higher than that of peri-implant inflammation,indicating that biomechanical balance at the implant bone tissue interface is crucial for long-term implant survival[28].The analysis of stress distribution and stress mode in three-dimensional finite element models can help improve the design of various implants.Arinc[29] used FEA to evaluate the stress effects of crown repair materials and structures on the implant and surrounding bone tissue, and analyzed the stress distribution of mandibular implant supported fixed dentures under oblique loading with different materials (cobalt chromium based ceramics, zirconium based ceramics, and zirconium reinforced polymethyl methacrylate resin) and different connector widths (2,3,and 4 mm). The results showed that cobalt chromium supported ceramics had the smallest stress on the implant. The implant material and connector width may affect the stress of cortical bone, cancellous bone, and implant.

6. Application of three-dimensional finite element analysis in the field of oral and maxillofacial surgery

The oral and maxillofacial regions are exposed parts of the human body, which are prone to damage. At the same time, due to the complex anatomical structure of the maxillofacial region, traditional research methods are time-consuming, consumable, and cannot be repeated, and the experimental research process is complex. In the field of oral and maxillofacial surgery, three-dimensional finite element analysis is mainly applied to simulate and analyze the forces on the jaw during the fracture process. In the maxillofacial region, mandibular angle fractures often result in postoperative infection and poor bone healing due to their high incidence of complications and complex surgical pathways. The stress analysis of a three-dimensional finite element model can further understand the relationship between steel nails and bone tissue during the rehabilitation process of patients. Three dimensional finite element analysis studies have shown that placing a single bicortical plate at the lower edge of the fracture area has better stability compared to placing a single tension band at the upper edge.Therefore, it is recommended in clinical practice to use a single tension band structure fixation as a minimally invasive method for treating fractures[30]. Real specimens of the jawbone are difficult to obtain, and three-dimensional finite element models can simulate real maxillofacial surgical structures, thereby reducing the difficulty of related

ISSN 2616-5791 Vol.4, Issue 10: 66-72, DOI: 10.25236/AJMHS.2023.041011

research.Huempfne Heierl et al.[31] confirmed that three-dimensional finite element models can be used to simulate human skull trauma and provide information on pathology and various fracture types. Bujt á r et al. [32]found that three-dimensional finite element analysis can also be used to construct models of the three developmental stages of the human mandible and analyze stress distribution. A study has found that when subjected to traumatic loads simultaneously, the three-dimensional finite element analysis model shows the highest stress value on the chin[33]. These results indicate that the level of stress is related to the position of stress and bone density, and have guiding significance for surgical operations.

7. Summary and Prospect

In clinical practice, analyzing the stress distribution of the oral cavity under different load conditions is crucial for evaluating the loss of teeth and restorations.Due to the complexity of tooth structure, the differences in mechanical and chemical properties between various tissue materials, and the complexity of tooth morphology and surrounding structural relationships, many experimental methods and methods cannot obtain accurate and reliable results[34]. There are currently two commonly used FEM models:2D modeling and 3D modeling.2D saves costs, simplifies materials to homogeneity and homogeneity, and modeling is simple and rough, which can be used for qualitative research.3D is more precise and can be used for quantitative research, and 3D modeling can restore some of the nonlinear and anisotropic characteristics of tissues, which can be more precise through technological development. Many studies have shown that the application of FEM in oral mechanics has become a mainstream analysis method for studying stress distribution. The rapid development of computer and modeling technology has also made FEM increasingly a very reliable and accurate method in the application of biomechanics.However, FEM also has its limitations. In clinical practice, the stress on tissues is dynamic and cyclic, while most studies on finite element models focus on simulating the static state of loading rather than actual clinical situations.Jongsma et al.[35]found in their research on the stress modes of fiber posts under different modeling methods that even if the model uses nonlinear modeling methods, which are already better than most similar studies, there is still a gap from reality, and the results obtained cannot be directly applied to clinical practice. In addition, setting the properties of the material to both anisotropy and non-uniformity can yield more accurate models,but requires more complex mathematical calculations.Therefore, how to establish a more accurate three-dimensional model has become a key issue in its application. Many high-tech technologies have been applied in the oral field, such as intraoral scanners, occlusal recorders, CAD/CAM systems, and FEM[36].The future breakthrough lies in the integration of these technology applications, such as using intraoral scanners to determine geometric characteristics, transferring data to CAD for treatment plan design, and then loading bite force through FEM to indicate whether there is excessive force, facilitating the improvement of plan design, and ultimately transforming from CAD design to CAM production process. The workload of modeling in three-dimensional finite element analysis accounts for 70% -80% of the entire workload. Reducing modeling time and simplifying FEM difficulty through new means is an important direction for development. If automated modeling can be achieved, it will achieve the idea of clinical specific modeling and greatly improve the application of FEM. The future application of three-dimensional finite element analysis should focus on model optimization and validation of bone tissue, temporomandibular joint, natural teeth, and dentures. At present, finite element analysis is more suitable for predicting situations where new methods and the clinical effects of these new methods can be confirmed in the future, or for comparing new instruments with traditional instruments. Predicting the similarities and differences of each group through finite element calculation can lay a solid foundation for further clinical research.

References

[1] Li HY, Li JY, Zou ZM, et al. Fracture simulation of restored teeth using a continuum damage mechanics failure model[J]. Dent Mater, 2011, 27(7): e125-e133.
[2] Dos Santos MB, Da Silva Neto JP, Consani RL, et al. Three-dimensional finite element analysis of stress distribution in peri-implant bone with relined dentures and different heights of healing caps[J]. J Oral Rehabil, 2011, 38(9):691-696.
[3] Gluskin AH, Peters CI, Peters OA. Minimally invasive endodontics: challenging prevailing paradigms [J]. Br Dent J, 2014, 216(6):347-353.
[4] Liu ZY, Zhao L, Yang LY, et al. Three dimensional finite element analysis of the effect of pulp opening and full crown restoration on the stress distribution of maxillary central incisors [J]. West China Journal of Stomatology, 2019, 37(6):642-647.

[5] Rodrigues YL, Mathew MT, Mercuri LG, et al. Biomechanical simulation of temporomandibular joint replacement (TMJ R) devices: a scoping review of the finite element method [J]. Int J Oral Maxillofac Surg, 2018, 47(8):1032-1042.
[6] Ozkurt-Kayahan Z, Turgut B, Akin H, et al. A 3D finite element analysis of stress distribution on different thicknesses of mineral trioxide aggregate applied on various sizes of pulp perforation [J]. Clin Oral Investig, 2020, 24(10):3477-3483.
[7] Kim S, Chen DZ, Park SY, et al. Stress analyses of retrograde cavity preparation designs for surgical endodontics in the mesial root of the mandibular molar:a finite element analysis-part II[J]. J Endod, 2020, 46(4):539-544.
[8] Ran SJ, Yang X, Sun Z, et al. Effect of length of apical root resection on the biomechanical response of a maxillary central incisor in various occlusal relationships [J]. Int Endod J, 2020, 53(1):111-121.
[9] Zhang H, Cui JW, Lu XL, et al. Finite element analysis on tooth and periodontal stress under simulated occlusal loads[J]. J Oral Rehabil, 2017, 44(7):526-536.
[10] Tuna M, Sunbuloglu E, Bozdag E. Finite element simulation of the behavior of the periodontal ligament: a validated nonlinear contact model[J]. J Biomech, 2014, 47(12):2883-2890.
[11] Ortún-Terrazas J, Cego□no J, Santana-Penín U, et al. Approach towards the porous fibrous structure of the periodontal ligament using micro-computerized tomography and finite element analysis[J]. J Mech Behav Biomed Mater, 2018, 79:135-149.
[12] Suzuki A, Masuda T, Takahashi I, et al. Changes in stress distribution of orthodontic miniscrews and surrounding bone evaluated by 3-dimensional finite element analysis[J]. Am J Orthod Dentofacial Orthop, 2011, 140(6):e273-e280.
[13] Nalbantgil D, Tozlu M, Ozdemir F, et al. FEM analysis of a new miniplate: stress distribution on the plate, screws and the bone[J]. Eur J Dent, 2012, 6(1):9-15.
[14] Lee RJ, Moon W, Hong C. Effects of monocortical and bicortical mini-implant anchorage on bone-borne palatal expansion using finite element analysi[J]. Am J Orthod Dentofacial Orthop, 2017, 151(5): 887-897.
[15] Liu QH. Finite element analysis of the movement of upper and lower central incisors with different attachments under invisible orthodontic treatment [D]. Tangshan: North China University of Science and Technology, 2018.
[16] Zeno KG, Mustapha S, Ayoub G, et al. Effect of force direction and tooth angulation during traction of palatally impacted canines:a finite element analysis[J]. Am J Orthod Dentofacial Orthop, 2020, 157(3):377-384.
[17] Nagendraprasad K, Mathew S, Shivamurthy P, et al. Displacement and periodontal stress analysis on palatally impacted canine-A finite element analysis [J]. Indian J Dent Res, 2019, 30 (5):788-793.
[18] Moga RA, Cosgarea R, Buru SM, et al. Finite element analysis of the dental pulp under orthodontic forces [J]. Am J Orthod Dentofacial Orthop, 2019, 155(4):543-551.
[19] Dejak B, M□tkowski A. Influence of anterior teeth inclination on the strength of incisors with post and cores [J]. Prosthodontics, 2020, 70(1):3-12.
[20] She YH, Zhang YW, Liu YX, et al. Three dimensional finite element analysis of the effect of tip coverage thickness on the stress distribution of premolars restored with all ceramic high inlay [J]. West China Journal of Stomatology, 2019, 37(6):636-641.
[21] Zhang ZP, Sornsuwan T, Rungsiyakull C, et al. Effects of design parameters on fracture resistance of glass simulated dental crowns[J]. Dent Mater, 2016, 32(3):373-384.
[22] Miura S, Kasahara S, Yamauchi S, et al. Three-dimensional finite element analysis of zirconia all-ceramic cantilevered fixed partial dentures with different framework designs[J]. Eur J Oral Sci, 2017, 125(3):208-214.
[23] Shahmiri R, Das R. Finite element analysis of implant-assisted removable partial dentures: Framework design considerations [J]. J Prosthet Dent, 2017, 118(2):177-186.
[24] Chen Y, Wang C, Huang YD, et al. Biomechanical evaluation of the natural abutment teeth in combined tooth-implant-supported telescopic prostheses: a three-dimensional finite element analysis[J]. Comput Methods Biomech Biomed Engin, 2017, 20(9):967-979.
[25] Vairo G, Sannino G. Comparative evaluation of osseointegrated dental implants based on platform-switching concept: influence of diameter, length, thread shape, and in-bone positioning depth on stress-based performance [J]. Comput Math Methods Med, 2013, 2013:1-15.
[26] Shigemitsu R, Yoda N, Ogawa T, et al. Biological-data-based finite-element stress analysis of mandibular bone with implant-supported overdenture [J]. Comput Biol Med, 2014, 54:44-52.
[27] Duyck J, R nold HJ, van Oosterwyck H, et al. The influence of static and dynamic loading on marginal bone reactions around osseointegrated implants: an animal experimental study[J]. Clin Oral Implants Res, 2001, 12(3):207-218.
[28] Ueda T, Kremer U, Katsoulis J, et al. Long-term results of mandibular implants supporting an

ISSN 2616-5791 Vol.4, Issue 10: 66-72, DOI: 10.25236/AJMHS.2023.041011

overdenture: implant survival, failures, and crestal bone level changes[J]. Int J Oral Maxillofac Implants, 2011, 26(2):365-372.

[29] Arinc H. Effects of prosthetic material and framework design on stress distribution in dental implants and peripheral bone:a threedimensional finite element analysis[J]. Med Sci Monit. 2018, 24:4279-4287.

[30] Kimsal J, Baack B, Candelaria L, et al. Biomechanical analysis of mandibular angle fractures [J]. J Oral Maxillofac Surg, 2011, 69 (12):3010-3014.

[31] Huempfner-Hierl H, Schaller A, Hemprich A, et al. Biomech-anical investigation of Naso-orbitoethmoid trauma by finite element analysis[J]. Br J Oral Maxillofac Surg, 2014, 52(9):850-853.

[32] Bujtár P, Sándor GK, Bojtos A, et al. Finite element analysis of the human mandible at 3 different stages of life[J]. Oral Surg Oral Med Oral Pathol Oral Radiol Endod, 2010, 110(3):301-309.

[33] Santos LS, Rossi AC, Freire AR, et al. Finite-element analysis of 3 situations of trauma in the human edentulous mandible[J]. J Oral Maxillofac Surg, 2015, 73(4):683-691.

[34] Cervino G, Fiorillo L, Arzukanyan AV, et al. Application of bioengineering devices for stress evaluation in dentistry:the last 10 years FEM parametric analysis of outcomes and current trends[J]. Minerva Stomatol, 2020, 69(1):55-62.

[35] Jongsma LA, Ir Nde J, Kleverlaan CJ, et al. Reduced contraction stress formation obtained by a two-step cementation procedure for fiber posts [J]. Dent Mater, 2011, 27(7):670-676.

[36] Reddy MS, Sundram R, Eid Abdemagyd HA. Application of finite element model in implant dentistry:a systematic review[J]. J Pharm Bioallied Sci, 2019, 11(Suppl 2): S85-S91.