Defence Life Science Journal, Vol. 9, No. 4, October 2024, pp. 293-300, DOI : 10.14429/dlsj.9.19439

Relevance of Finite Element Method in Dentistry: Finite with Infinite Possibilities

Aishwarya Pandey[*], Lt Col Parvinder Sharma[$] and Gayathrie B[$]
[*]*Faculty of Dental Science, Institute of Medical Sciences, Banaras Hindu University, Varanasi - 221 005, India*
[$]*DRDO - Institute of Nuclear Medicine and Allied Sciences, Timarpur, Delhi – 110 054, India*
[*]*E-mail: aishwarya22.ace@gmail.com*

ABSTRACT

This narrative review gives an insight into the Finite Element Method (FEM) and its principles comprehensible from a dentist's point of view along with its applications in different specialties of dentistry. FEM is a method for simulating the behavior of a physical system mathematically. For this, a complex structure is broken down into several smaller components (elements) while still retaining its original characteristics. Subsequently, differential equations are used to explain and solve each component. With the advent of FEM, it became easier to comprehend various elements of oral biomechanics. Biomechanical studies designed in partnership with skilled computer engineers and experienced clinicians not only provide a better insight into the mechanisms of stress distribution but also help prepare customized treatment plans that cater to the requirements of individual patients.

Keywords: Finite element method; Biomechanical analysis; Advancements in dentistry

NOMENCLATURE

FEM	: Finite element method
FEA	: Finite element analysis
CAD	: Computer-aided design
CAM	: Computer-aided manufacturing
CT:	: Computed tomography
PDL	: Periodontal ligament
VRF	: Vertical root failure
PSI	: Patient-specificimplant
IOFF	: Isolated orbital floor fractures
ICR	: Instantaneous center of rotation
SSC	: Stainless steel crowns
TDI	: Traumatic dental injury

1. INTRODUCTION

With the introduction of several technological advances, such as nanoscience technology, regenerative biomaterials and other bio-engineering technologies, whose applications in the field of dentistry are developing dramatically, a new age in dentistry has begun. The detailed understanding of the nature and distinctive characteristics of human hard and soft tissues and numerous tools and techniques used in dentistry at a microscopic and ultra-structural level have been made possible by the convergence of biological and engineering sciences through several technological advancements. The Finite Element Method (FEM) is one such development in the world of bioengineering. The oral cavity is a sophisticated biomechanical system. Mechanical tests have been performed to evaluate the mechanical characteristics such as strength, hardness, toughness, fracture resistance, etc. of dental structures, restorative materials, and implants, but these tests do not reveal details about how the structures under study behave internally. When an external force acts upon a structure, it leads to internal deformation and strains to develop within it. Structural collapse could happen if stresses rise above the elastic limit and become excessive. The failure of the prosthesis, bone remodeling, and the type of tooth movement are all determined by stress. It is difficult to comprehend why and when a failure process begins in complex systems, how the stresses affect the prosthesis and stomatognathic system, or how to directly assess these pressures. With the advent of Finite Element Method (FEM) in dentistry, it became easier to comprehend various elements of oral biomechanics. FEM has been widely recognised as an accurate, non-invasive method for biomechanical analysis and the impact of physical forces on the functioning of different biological systems. It makes it possible to define the physical attributes of anatomical craniofacial structures along with easy visualisation of superimposed elements[1]. It also allocates stress sites that can be measured, with determination of the location, dimensions, and direction of any applied force[2]. Furthermore, it is highly reproducible as it doesn't alter the physical characteristics of the materials being studied[2].

The FEM came into existence in 1956, primarily to be used in the field of aerospace engineering[3]. It was initially employed in dentistry as an alternative for photoelasticity testing in the 1970s[4].and Weinstein

Received : 04 September 2023 Revised : 25 August 2024
Accepted:10 September 2024 Online published :26 September 2024

pioneered its use in implant dentistry in 1976[5]. Thereafter, it has been extensively used in various specialties. FEM has undergone significant advancements since its inception to enhance sensitivity and specificity in various scientific applications. These improvements include the development of more refined element formulations capable of capturing complex material behaviors and geometric features, adaptive mesh refinement techniques that allow for local refinement in regions of interest, utilisation of high-performance computing for larger and more detailed simulations, integration of multiphysics and multiscale modeling approaches for comprehensive analysis, and validation and verification processes to enhance model accuracy[1].

This narrative review aims to give an insight into the FEM concept and principles comprehensible from a dentist's point of view along with its applications in dentistry(Table 1).

1.1 Decoding FEM

There are three ways to address an engineering problem, which includes experimental, numerical, and analytical. FEM falls under the category of numerical approach for investigating designs, which involves simulating a structure on a computer, subjecting it to a command, and evaluating the results[3].

FEM is a method for simulating the behavior of a physical system mathematically. For this, a complex structure is broken down into several components (elements) while still retaining its original characteristics[3].This process of breaking down a complex structure for its biomechanical analysis is called finite element modeling. Using mathematical models chosen as per the facts under investigation, differential equations are used to explain and solve each of these components. This process of analysing the model under various stress conditions is called Finite Element Analysis (FEA).
Fundamental stages in FEM include- Pre-processing, processing, and post-processing[6] (Fig. 1).

Table 1. Application of FEM across various specialties in dentistry

Specialty	Application of FEM
Orthodontics	- Prediction of tooth movement and bone remodeling patterns - Assessment of biomechanical effects of orthodontic forces on teeth and surrounding structures - Simulation of stress distribution in periodontal ligament and alveolar bone during orthodontic treatment - Optimization of treatment plans for complex malocclusions - Evaluation of biomechanical performance of orthodontic appliances and treatment modalities - Study of orthodontic relapse and long-term stability of treatment results - Assessment of anchorage mechanics and stability during orthodontic treatment
Oral Implantology	- Design and optimization of dental implant shapes, sizes, and surface characteristics - Assessment of biomechanical stability and osseointegration of dental implants - Evaluation of stress distribution in surrounding bone and implant-abutment complex - Comparison of different implant systems - Optimization of implant placement protocols and surgical techniques - Assessment of factors influencing implant success and longevity - Investigation of peri-implant bone loss and implant-related complications
Oral and Maxillofacial Surgery	- Determination of those parts of the craniofacial skeleton that are most susceptible to fracture injury - Evaluation of the mechanical resilience of various reconstructive techniques - Development and optimization ofpatient-specific implants - Assessment of bone healing and remodeling processes - Design and optimization of patient-specific implants (PSIs) for craniofacial reconstruction
Periodontics	- Simulation of periodontal tissue response to mechanical and chemical stimuli - Assessment of stress distribution and biomechanical behavior of periodontal ligament and alveolar bone - Assessment of the impact of various occlusal forces, the pattern of stress distribution experienced by the splinted teeth along with a comparative analysis of different splinting materials. - Optimization of treatment plans for periodontal disease and gingival recession
Prosthodontics	- Design and optimization of dental prostheses (e.g., crowns, bridges, dentures) - Assessment of stress distribution and biomechanical behavior of prosthetic restorations - Assessment of biomechanics of maxillary obturator prostheses - Study the impact of occlusal interference on dental prosthesis
Conservative and Endodontics	- Analysis of stress distribution and fracture resistance in endodontically treated teeth - Identification of the components that affect root fracture susceptibility - Prediction of the effects of different root canal preparation techniques, rotary instruments,and root anatomies for further assessment of fracture possibilities - Evaluation of root canal instrumentation techniques and materials
Pedodontics	- Assessment of biomechanical response of primary and permanent teeth to orthodontic forces - Simulation of stress distribution in pediatric craniofacial structures during growth and development - Evaluation of the post-traumatic luxation and avulsion patterns

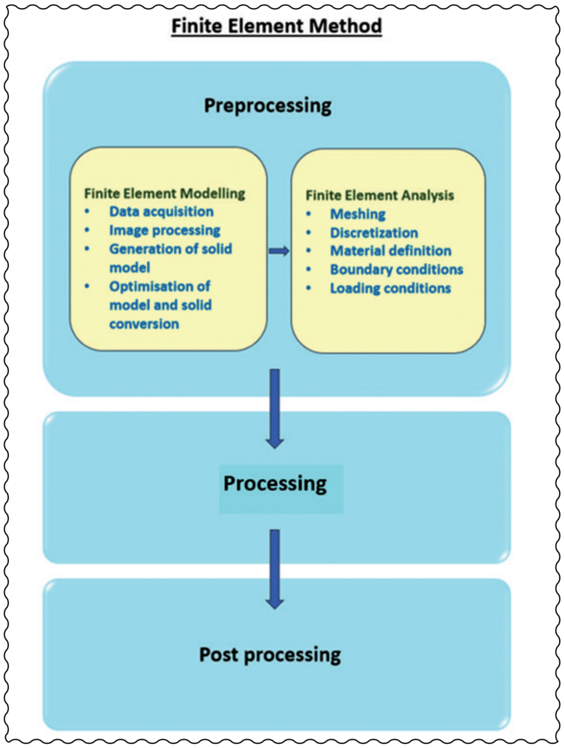

Figure 1. Fundamental stages in finite element method

Pre-processing stage

It includes both the modeling and the analysis phase. The modeling phase comprises data acquisition which is either via CAD (Computer-aided Design) or CT (Computed tomography) to create a 2-dimensional or 3-dimensional model (Fig. 2a), followed by optimisation of the model and solid conversion (Figure 2b). For analysis, the model is exported to FEA software (e.g., Abaqus Explicit, Ansys, Hypermesh, Nastran-Patran, etc.), and is visually simulated in a mesh format that specifies its layout and geometry[3].In a procedure called discretisation, the mesh is further broken down into smaller geometrical units called elements, which are linked at points known as nodes (Fig. 2c). Subsequently, material properties are defined to retain the original attributes of the structure, followed by the loading conditions that comprise the forces acting on the structure. Lastly, the model is subjected to certain constraints called boundary conditions to reduce the complexity of the analysis.

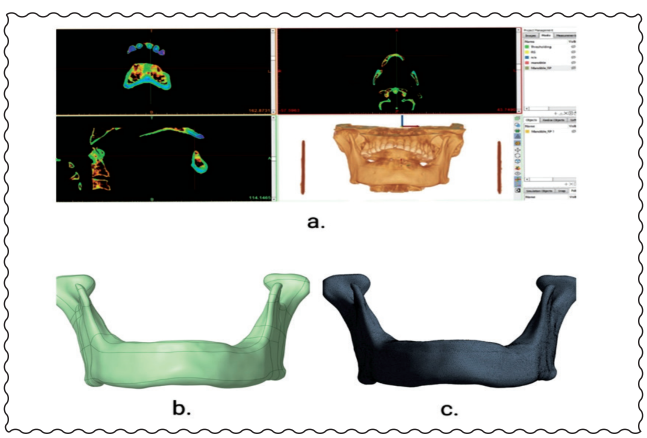

**Figure 2. a. Construction of the model from CT images.
b. Optimization of 3-D Model
c. Meshing**

Processing stage

During this stage, complex calculations such as matrix formulations, inversions, and multiplications are performed.

Post-processing stage

In the post-processing stage, the result of the concluding design is further validated and improved as per the requirements of the study[6].

The fact that stress and strain cannot be directly quantified in human tissues in response to external forces (due to ethical considerations in conducting the research), makes this method of immense importance in assessing the physical attributes of different materials and procedures in the human body. The generated findings may then be examined using visualisation tools within the FEM framework to explore an assortment of parameters and comprehend the implications of the study[3].

1.2 Merits and Demerits of FEM

FEM provides several benefits over research using actual models (Fig. 3). There are no ethical issues, the tests can be repeated, and the research designs may be altered to suit the situation.

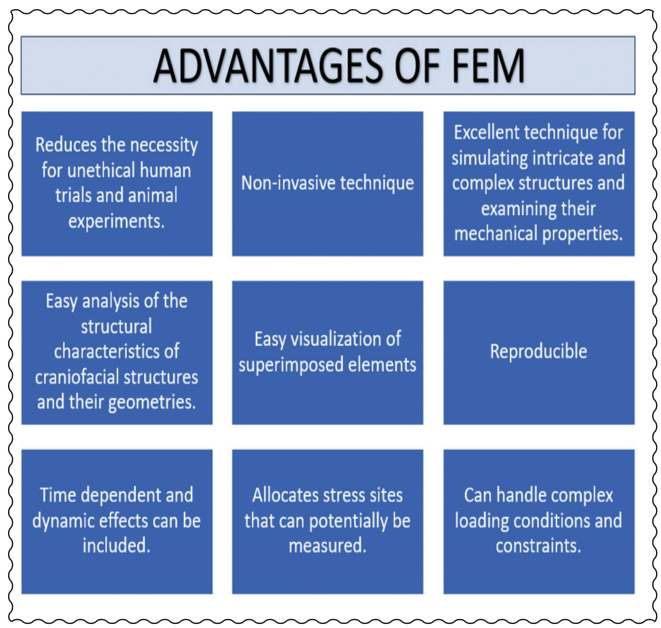

Figure 3. Advantages of FEM

FEM also comes with certain limitations. The clinical state may not be precisely duplicated in this computerised in vitro investigation. Contrary to actuality, materials' mechanical properties are typically represented as being isotropic and linearly elastic. Additionally, the stress analysis is performed using static loads, which again differs from the real-world scenario.

2. APPLICATIONS OF FEM IN DENTISTRY

2.1 Finite Element Method in Implant Dentistry

A notable landmark success of FEM was a simulation of dental implant biomechanics, which has helped optimise implant design and placement for better osseointegration and long-term stability. FEM has been used to explore dental implant designs, the structure and material of the superstructure, the behavior of implant under various stresses, and the effects it has on the bone in the vicinity[7, 8]. To limit crestal bone loss and failure of implants as well as for enhancing the durability and lifetime of implant designs, it is essential to comprehend the biomechanical behavior of alveolar bone and dental implants. Therefore, the scope of biomechanical studies in the domain of implant dentistry has significantly increased.

The bone quality and quantity, the kind of prosthesis, loading conditions, bone-implant contact area, implant dimensions, and the form and features of the surface of an implant, act together in determining the stress distribution patterns. Atmaram and Mohamed evaluated stress dispersion patterns in a single-tooth implant using FEM for a better understanding of the implications of implant geometry and its elastic attributes, varying implant dimensions, and inclusion of a pseudo-periodontal ligament[9,10].FEM investigations of osseointegrated implants show that under centric loading, the greatest stress concentration occurs in the contact region of the implant in cortical bone, and at the implant's apex in the cancellous bone[11,12].For full mouth rehabilitation, it has been observed in several FEM studies that there is better stress distribution on implants and in bone when the number of implants is increased[13, 14].Biomechanical studies also indicated that the cantilever plays a huge role in affecting the stress distribution as implants incurred an increased amount of stress proportional to the cantilever length[15,16].

2.2 Finite Element Method in Endodontics

The fundamental objectives of root canal therapy are to treat the infection while safeguarding the tooth's health and functionality. Though the survival rate of primary root canal treatment accounts for more than 90 %, there are some instances where the longevity of a root canal-treated tooth is doubtful such as compromised periodontal apparatus, non-restorable caries, iatrogenic errors, and vertical root fractures[17,18].Vertical Root Failure (VRF) is a more frequently occurring cause behind the failure of a restored root canal-treated tooth. In addition to its challenging diagnosis, management frequently necessitates very drastic measures, such as extraction or root amputation[19]. FEM can be employed to identify the components that affect fracture susceptibility[19,20]. Studies have shown that FEA models accurately predict the actual fracture pattern during fracture strength testing, supporting the viability of Vertical Root Failure (VRF)[21,22].

FEM can also predict the effect of different root canal preparation techniques for further assessment of fracture possibilities[19]. Cheng et al. investigated the stress distribution on teeth with curved canals that had undergone endodontic treatment under different pressures and found that when severe compaction forces (50 N) were imposed, the warm vertical compaction approach was likely to cause root fractures[19]. FEM models have been created to quantitatively analyse the stress distribution of teeth post root canal treatment to assess the relative role of geometrical variables to tooth fracture. According to Ricks-Williamson et al., radicular stresses generated in their finite element model were directly proportional to the canal widths[23]. Sathorn *et al.,* in their finite element study observed that the interplay between dentin thickness, root surface curvature, canal size, and form have a collaborative role in influencing fracture susceptibility along with fracture pattern[24].

There has been an increased incidence of root fractures with the advent of NiTi rotary instruments[19]. *Kim et al.,* assessed the geometrical variations among three NiTi instruments which affect the stress distribution patterns in bending and torsional situations and found that the most adaptable of the three file models was ProFile, with a U-shaped cross-section while the ProTaper was the stiffest file model with a convex triangular cross-section[25].

2.3 Finite Element Method and Periodontics

Continuous and unchecked attachment loss around the tooth may ultimately lead to tooth mobility and eventual tooth loss. Increased mobility not only affects the normal functioning of the affected teeth but also raises discomfort and aesthetic concerns. In such cases of compromised periodontium,it is crucial to determine if splinting would avert additional bone loss and for this, the biomechanical effects of splints on bone become a crucial consideration when choosing a suitable treatment strategy. FEM can be employed in such cases to assess the impact of various occlusal forces, the pattern of stress distribution experienced by the splinted teeth along with a comparative analysis of different splinting materials.

In comparison to vertical loading, oblique loading led to greater stress values on periodontal tissues, and splints[26]. Amid et al. in their finite element study observed that the teeth close to the splint's central axis experienced lesser stress while the teeth farther from it encountered a greater magnitude of stress[26]. In a similar study by Galohda *et al.,* the authors found that the metal-reinforced composite had better stress distribution and was superiortoother splinting materials[27]. According to Liu *et al.,* oblique loading led to greater stress values on periodontal tissues, and splints when compared to vertical loading[28].

2.4 Finite Element Method in Oral and Maxillofacial Surgery

Oral and maxillofacial surgery mandates basic comprehension of the behavior of bone in response to biomechanical perturbations, fracture mechanisms, and the mechanical attributes of various osteosynthesis materials. FEM can be utilised in trauma surgery to determine those parts of the craniofacial skeleton that are most susceptible to fracture injury[29]. Using FEA, de Mello Santos et al. examined the stress distributions caused by traumatic loads that were exerted on three regions- symphyseal, parasymphyseal, and mandibular body in elderly edentulous mandibles[30]. They observed that traumatic load in the symphyseal area led to high levels of stress in the mandibular neck while load in the parasymphyseal area localised the stress to the mental foramen. The ramus, angle, and mandibular body all experienced significant stress as a result of the trauma to the mandibular body. FEM was also used to study more complicated injuries, for instance, Isolated Orbital Floor Fractures (IOFF) and zygomatic bone fractures, that are often encountered in contact sports[31,32]. Pramana *et al.,* did a FEM to evaluate the mechanical resilience of six big polymethylmethacrylate reconstructions for rehabilitation of cranial defects and found that for small-sized defects it was easier to build an even distribution of reconstruction plates but for extremely big-sized defects with complex geometries, it was not the case[33]. Hence, it is advantageous to use multi-design computational studies for choosing the best course of action for a complex clinical situation. FEM has been instrumental in analysing the performance of patient-specific implants (PSIs). It plays a crucial role in the development and optimisation of PSIs. Through iterative simulation and analysis, FEM allows clinicians and engineers to evaluate the biomechanical performance of PSIs, predicting factors such as stress distribution, deformation, and stability under physiological loading conditions[34,35].

2.5 Finite Element Method in Prosthodontics

The longevity of restoration in the oral environment is largely dictated by the restoration's resistance to fracture. Restorations with a high resistance to fracture have better survival rates when subjected to chewing forces which highlights the importance of biomechanical factors in influencing the prosthetic success[36]. FEM is used in prosthetic dentistry for a better understanding of the denture characteristics, individual dental crowns, and their physical and chemical attributes. FEA simulation models of maxillary resection can be used to study the biomechanics of maxillary obturator prostheses.

De Saussa and Mattos observed in their FEM study that as the area of bone support, the total number of implants, and clips decreased, the amount of dislodgment and instability of the obturator prosthesis increased[37].The gingival mucosa, cancellous, and cortical bone, were also subjected to compressive force and this stress grew as the area of bone support, the number of implants, and the quantity of clips in the bar retention system shrank. Schmid *et al.,* used FEM to study the impact of occlusal interference on dental prosthesis[38]. Peak tensile stress was seen to increase with increased occlusal interference in close vicinity to the occlusal contact locations. Also, the periodontal ligament's deformation serves as the mechanism for absorbing a significant amount of occlusal stress. Coelho *et al.,* did a FEM study to compare the influence of standard chairside fabrication techniques with CAD-CAM on the occurrence of flaws and the mechanical characteristics of an interim dental prosthesis and found that the strength of interim partial prosthesis made with CAD-CAM was greater than that with conventional technique[39].

2.6 Finite Element Method in Orthodontics

When a tooth is subjected to force, the periodontium is displaced in a variety of ways causing orthodontic tooth movement. In response to stress on the periodontal ligament, cells resorb as well as deposit the alveolar bone, ultimately resulting in tooth movement. FEM has been instrumental in the realm of orthodontics, as it has been used to simulate the biomechanical effects of orthodontic forces on teeth and surrounding structures, aiding in successful treatment planning and optimisation.

The primary applications of this technology are to study tooth mobility, skeletal anchorage, and bracket–enamel interaction[40,41,42].Using the FEM, Williams, and Edmundson investigated the location of the Instantaneous Center of Rotation (ICR) of a maxillary central incisor and found that the rotational center is unaffected by the PDL's elastic characteristics[43]. Although the ICR's location is influenced by the point of loading, it is not reliant on the load per se. Rudolph et al. conducted a FEM study to ascertain the displacement and stress distribution of 5 distinct load configurations on a maxillary central incisor[44]. The authors demonstrated that stresses from pure intrusive, extrusive, and rotating forces were localised at the root's apex. The alveolar crest was the site of the main tension caused by a tipping force. With the use of the FEM, multiple studies have examined how orthodontic stresses affect the craniofacial complex[45,46]. FEM studies have shown that other characteristics, such as wire type, width of the bracket, wire length, and degree of misalignment, have a greater impact on the generated torquing moment than does bracket design[47].

2.7 Finite Element Method in Pediatric Dentistry

The primary dentition acts as the best space maintainer and plays a crucial role in preserving the arch integrity. However, due to the prevalence of dental caries in children, it becomes necessary to preserve the tooth's functionality by performing restorative procedures. To reinforce the tooth's strength, certain restorative materials are employed such as amalgam, glass ionomer cements, and composites, along with stainless-steel and zirconia crowns[48].

Prabhakar *et al.,* compared the efficiency of prefabricated zirconia crowns with the gold standard Stainless Steel Crowns (SSC) for restoration of deciduous teeth with the help of FEM and found that grossly damaged teeth treated with prefabricated zirconia crowns can sustain stress better than teeth restored with SSC, even at the highest levels of physiological masticatory forces[48]. While the primary purpose of luting cement is to hold the crown in place, it also serves as a shock absorber, distributes weight to the supporting tooth structures, and creates an integrated structure to respond as a single unit to occlusion force.

Employing FEM, Waly *et al.* investigated the impact of employing various cement types beneath pediatric SSC around mandibular second primary molars[49].The authors found that utilising more rigid cement material causes less stress on the crown's body and more stress on the tooth structure. Traumatic Dental Injuries (TDIs), which account for 5 % of all injuries for which patients seek medical attention, are quite common in young children, school-age children, and adults. FEM is often employed to evaluate the post-traumatic luxation and avulsion patterns along with biomechanical analysis of splints[50].

3. CONCLUSION

To more accurately reflect the clinical setting, futuristic FEM-based research should concentrate on analysing stress distributions during dynamic loading circumstances and real-time simulations. Biomechanical studies designed in partnership with skilled computer engineers and experienced clinicians not only provide a better insight into the mechanisms of stress distribution but also help in designing individualised treatment plans that cater to the requirements of every patient.

REFERENCES

1. Sun, J.; Jiao, T.; Tie, Y. & Wang, D.M. Three-dimensional finite element analysis of the application of attachment for obturator framework in unilateral maxillary defect. *Journal of oral rehabilitation*, 2008, **35**, 695–699. doi: 10.1111/j.1365-2842.2007.01801.x.
2. Gao, J.; Xu, W. & Ding, Z. 3D finite element mesh generation of complicated tooth model based on CT slices. *Computer methods and programs in biomedicine*, 2006, **82**, 97–105. doi: 10.1016/j.cmpb.2006.02.008
3. Shivakumar, S.; Kudagi, V.S. & Talwade, P. Applications of finite element analysis in dentistry: A review. *J. Int. Oral Health,* 2021, **13**, 415-422. doi: 10.4103/JIOH.JIOH_11_21
4. Thresher, R.W. & Saito, G.E. The stress analysis of human teeth. *J. Biomech,* 1973, **6,** 443-449. doi: 10.1016/0021-9290(73)90003-1
5. Weinstein, A.M.; Klawitter, J.J.; Anand, S.C. & Schuessler, R. Stress analysis of porous rooted dental implants. *J. Den.t Res.,*1976, **55**, 772-777. doi: 10.1177/00220345760550051001
6. Shetty, P.; Hegde, A.M. & Rai, K. Finite element method: An effective research tool for dentistry. *J. Clin. Pediatr Dent*, 2010, **34**, 281-285. doi: 10.17796/jcpd.34.3.yu44334815546435
7. Reddy, M.S.; Sundram, R.; Eid, & Abdemagyd H.A. Application of finite element model in implant dentistry: A systematic review. *J. Pharm. Bioall Sci*, 2019, **11**, S85-91. doi: 10.4103/JPBS.JPBS_296_18
8. Geng, J.P.; Tan, K. B.; & Liu, G.R. Application of finite element analysis in implant dentistry: A review of the literature. *J. Prosthet Dent.,* 2001, **85**, 585-598. doi: 10.1067/mpr.2001.115251
9. Atmaram, G.H.; & Mohammed, H. Stress analysis of single-tooth implants. I. Effect of elastic parameters and geometry of implant. *Implantologist*, 1983-1984, **3**, 24-29. doi: 10.3109/10731197909119376
10. Mohammed, H.; Atmaram, G.H.; & Schoen, F.J. Dental implant design: A critical review. *J. Oral Implantol*, 1979, **8**, 393-410.
11. Ma Xuanxiang, L.T. Single implant prosthesis. Newly developed technology of prosthetic dentistry in China in 1990s, edited by Geng Jianping.Chengdu: Sichuang science and technology publishing house, China, 1998, pp.1-20.
12. Clift, S.E.; Fisher, J.; & Watson, C.J. Finite element stress and strain analysis of the bone surrounding a dental implant: Effect of variations in bone modulus. *Proc. Inst. Mech. Eng.*, 1992, **206**, 233-241. doi: 10.1243/PIME_PROC_1992_206_295_02
13. ÖzdemirDoğan, D.; Polat, N.T.; Polat, S.; Şeker, E. & Gül, E.B. Evaluation of "all-on-four" concept and alternative designs with 3D finite element analysis method. *Clin. Implant Dent. Relat. Res.*, 2014, **16**, 501–510. doi: 10.2478/bjdm-2021-0004
14. Pandey, A.; Durrani, F.; Rai, S.K.; Singh, N.K.; Singh, P.; Verma, R. & Anjum M. Comparison between all-on-four and all-on-six treatment concepts on stress distribution for full-mouth rehabilitation using three-dimensional finite element analysis: A biomechanical study. *Journal of Indian Society of Periodontology*, 2023, **27**, 180-188. doi: 10.4103/jisp.jisp_278_22
15. Bevilacqua, M.; Tealdo, T.; Menini, M.; Pera, F.; Mossolov, A.; Drago, C. & Paolo P. The influence of cantilever length and implant inclination on stress distribution in maxillary implant-supported fixed dentures. *J. Prosthet Dent*, 2011, **105**, 5–13. doi: 10.1016/S0022-3913(10)60182-5
16. Rubo, J.H. & Capello Souza, E.A. Finite-element analysis of stress on dental implant prosthesis. *Clin. Implant Dent. Relat Res.*, 2010, **12**, 105–113. doi:10.1111/j.1708-8208.2008.00142.x
17. De Chevigny, C.; Dao, T.T.; Basrani, B.R.; Marquis, V.; Farzaneh, M.; Abitbol, S. & Friedman, S. Treatment outcome in endodontics: The toronto study-phase 4: Initial treatment. *J. Endod.*, 2008, **34**, 258-263. doi: 10.1016/j.joen.2007.10.017
18. Touré, B.; Faye, B.; Kane, A.W.; Lo, C.M.; Niang, B. & Boucher, Y. Analysis of reasons for extraction of endodontically treated teeth: A prospective study. *J. Endod,* 2011, **37**, 1512-1515. doi: 10.1016/j.joen.2011.07.002
19. Hu, T.; Cheng, R.; Shao, M.; Yang, H.; Zhang, R. & Gao, Q. *et al.,* Application of finite element analysis in root canal therapy [Internet]. finite element analysis. Sciyo. [Accessed on 2023 May 03; Cited

on 2023 Jun 10]. Available from:
doi: 10.5772/9994

20. Williams, K.R. & Edmundson, J.T.A finite element stress analysis of an endodontically restored tooth. *Eng. Med.*, 1984, **13**, 167-173.
doi: 10.1243/emed_jour_1984_013_043_02
21. Lertchirakarn, V.; Palamara, J.E. & Messer, H.H. Finite element analysis and strain-gauge studies of vertical root fracture. *J. Endod,* 2003, **29**, 529-534.
doi: 10.1097/00004770-200308000-00009
22. Sathorn, C.; Palamara, J.E. & Messer, H.H.A. Comparison of the effects of two canal preparation techniques on root fracture susceptibility and fracture pattern. *J. Endod,* 2005, **31**, 283-287.
doi: 10.1097/01.don.0000140580.03121.61
23. Ricks-Williamson, L.J.; Fotos, P.G.; Goel, V.K.; Spivey, J.D.; Rivera, E.M. & Khera, S.C.A three-dimensional finite-element stress analysis of an endodontically prepared maxillary central incisor. *Journal of Endodontics*, 1995, **21**, 362-327.
doi: 10.1016/S0099-2399(06)80971-4
24. Sathorn, C.; Joseph, E.A. & Harold, H. Effect of root canal size and external root surface morphology on fracture susceptibility and pattern: A finite element analysis. *Journal of Endodontics*, 2005, **31**, 288-292.
doi: 10.1097/01.don.0000140579.17573.f7
25. Kim, H.C.; Cheung, G.S.; Lee, C.J.; Kim, B.M.; Park, J.K.& Kang, S.I. Comparison of forces generated during root canal shaping and residual stresses of three nickel–titanium rotary files by using a three-dimensional finite-element analysis. *Journal of Endodontics*, 2008, **34**, 743-747.
doi: 10.1016/j.joen.2008.03.015
26. Amid, R.; Kadkhodazadeh, M.; Dehnavi, F. & Brokhim, M. Comparison of stress and strain distribution around splinted and non-splinted teeth with compromised periodontium: A three-dimensional finite element analysis. *J. Adv. Periodontol Implant Dent.*, 2018, **20**, 35-41.
doi: 10.15171/japid.2018.007
27. Durrani, F.; Galohda, A.; Rai, S.K.; Singh, N.K.; Verma, R. & Yadav, D.S. *et al.,* Evaluation and comparison of stress distribution around periodontally compromised mobile teeth splinted with different materials: Three-dimensional finite element analysis. *Indian J. Dent. Res.*, 2019, **30**, 97-101.
doi: 10.4103/ijdr.IJDR_250_17
28. Liu, Y.; Fang, M.; Zhao, R.; Liu, H.; Tian M. & Zhong, S., *et al.,* Effects of periodontal splints on biomechanical behaviors in compromised periodontal tissues and cement layer: 3D finite element analysis. *Polymers (Basel)*, 2022, **14**, 2835.
doi: 10.3390/polym14142835
29. Perestrelo, P.; De Oliveira, J.; Noritomi, P. & Da Silva, J. Application of a virtual cranial model in trauma simulation. *Procedia CIRP*, 2016, **49**, 19–22.
doi: 10.1016/j.procir.2015.07.022
30. Santos, L.S.; Rossi, A.C.; Freire, A.R.; Matoso, R.I.; Caria, P.H. & Prado, F.B. Finite-element analysis of 3 situations of trauma in the human edentulous mandible. *J. Oral MaxillofacSurg.*, 2015, **73**, 683-691.
doi: 10.1016/j.joms.2014.10.014
31. Wanyura, H.; Kowalczyk, P.; SmolczykWanyura, D.; Stopa, Z. & Bossak, M. Finite element analysis of external loads resulting in isolated orbital floor fractures. *J. Stoma*, 2011, **64**, 476–489.
doi: L10.5114/ninp.2012.30267
32. Schaller, A.; Voigt, C.; Huempfner-Hierl, H.; Hemprich, A. & Hierl, T. Transient finite element analysis of a traumatic fracture of the zygomatic bone caused by a head collision. *Int. J. Oral MaxillofacSurg*, 2012, **41**, 66–73.
doi: 10.1016/j.ijom.2011.09.004
33. Ridwan-Pramana, A.; Marcián, P.; Borák, L.; Narra, N.; Forouzanfar, T. & Wolff, J. Finite element analysis of 6 large PMMA skull reconstructions: A multi-criteria evaluation approach. *PLoS One,* 2017, 12:e0179325.
doi: 10.1371/journal.pone.0179325
34. Akshay, Kumar, J.; Kode, J.P.; Shivraj, Narayan, Y. & Satyanarayana, B. Finite element analysis of a patient specific maxillofacial implant. *International Journal of Engineering Trends and Technology,* 2022, **70**, 354-361.
doi: 10.14445/22315381/IJETT-V70I9P235
35. Vignesh, U.; Mehrotra, D.; Bhave, S.M.; Katrolia, R. & Sharma, S. Finite element analysis of patient-specific TMJ implants to replace bilateral joints with simultaneous correction of facial deformity. *Journal of oral biology and craniofacial research*, 2020, **10**, 674–679.
doi: 10.1016/j.jobcr.2020.07.013J
36. Bramanti, E.; Cervino, G.; Lauritano, F.; Fiorillo, L.; D'Amico, C. & Sambataro, S. *et al.,* FEM and von mises analysis on prosthetic crowns structural elements: Evaluation of different applied materials. *Scientific World Journal,* 2017, 1029574.
doi: 10.1155/2017/1029574
37. De Sousa, A. A. & Mattos, B.S. Finite element analysis of stability and functional stress with implant-supported maxillary obturator prostheses. *J. Prosthet Dent*, 2014, **112**, 1578-1584.
doi: 10.1016/j.prosdent.2014.06.020
38. Schmid, A.; Strasser, T. & Rosentritt, M. Finite element analysis of occlusal interferences in dental prosthetics caused by occlusal adjustment. *Int. J. Prosthodont*, 2021, **36**, 436–442.
doi: 10.11607/ijp.7178.
39. Coelho, C.; Calamote, C.; Pinto, A.C.; Esteves, J.L.; Ramos, A. & Escuin, T., *et al.,* Comparison of CAD-CAM

and traditional chairside processing of 4-unit interim prostheses with and without cantilevers: Mechanics, fracture behavior, and finite element analysis. *J. Prosthet Dent.*, 2021, **125**, 543. e1-543.e10.
doi: 10.1016/j.prosdent.2020.11.007

40. Kojima, Y. & Fukui, H. Numerical simulation of canine retraction by sliding mechanics. *Am J. Orthod Dentofacial Orthop*, 2005, **127**, 542–551.
doi: 10.1016/j.ajodo.2004.12.007
41. Suzuki, A.; Masuda, T.; Takahashi, I.; Deguchi, T.; Suzuki, O. &Takano Yamamoto, T. Changes in stress distribution of orthodontic miniscrews and surrounding bone evaluated by 3-dimensional finite element analysis. *Am. J. Orthod Dentofacial Orthop*, 2011, **140**, 273–280.
doi:10.1016/j.ajodo.2011.06.025
42. Lin, L.; Huang, S.F.; Tsai, H. C.& Chang, W.J. Finite element submodeling analyses of damage to enamel at the incisor enamel/adhesive interface upon de-bonding for different orthodontic bracket bases. *J. Biomech*, 2011, **44**, 134–142.
doi: 10.1016/j.jbiomech.2010.08.038
43. Williams, K.R. & Edmundson, J.T. Orthodontic tooth movement analysed by the finite element method. *Biomaterials*, 1984, **5**, 347-351.
doi: 10.1016/0142-9612(84)90033-4
44. Rudolph, D.J.; Willes, P.M. &, Sameshima, G.T.A finite element model of apical force distribution from orthodontic tooth movement. *Angle Orthod*, 2001, **71**, 127-131.
doi: 10.1043/0003-3219(2001)071<0127:AFEMOA>2.0.CO;2
45. McGuinness, N.; Wilson, A.N.; Jones, M.; Middleton, J. & Robertson, N.R. Stresses induced by edgewise appliances in the periodontal ligament-a finite element study. *Angle Orthod*, 1992, **62**, 15-22.
doi: 10.1043/0003-3219(1992)062<0015:SIBEAI>2.0.CO;2
46. Shaw, A.M.; Sameshima, G.T. & Vu, H.V. Mechanical stress generated by orthodontic forces on apical root cementum: A finite element model. *OrthodCraniofac Res.*, 2004, **7**, 98-107.
doi: 10.1111/j.1601-6343.2004.00285.x
47. Huang, Y.; Keilig, L.; Rahimi, A.; Reimann, S. & Bourauel, C. Torque capabilities of self-ligating and conventional brackets under the effect of bracket width and free wire length. *OrthodCraniofac Res.*, 2012, **15**, 255-262.
doi: 10.1111/j.1601-6343.2012.01553.x
48. Prabhakar, A.R.; Chakraborty, A.; Nadig, B. & Yavagal, C. Finite element stress analysis of restored primary teeth: A comparative evaluation between stainless steel crowns and preformed zirconia crowns. *Int. J. Oral Health Sci.*, 2017, **7**, 10-15.
doi: 10.4103/ijohs.ijohs_18_17
49. Waly, A.S.; Souror, Y.R.; Yousief, S.A.; Alqahtani, W.M.S. & El-Anwar, M.I. Pediatric stainless-steel crown cementation finite element study. *Eur. J. Den.t*, 2021, **15**, 77-83.
doi: 10.1055/s-0040-1715915
50. Vilela, A.B.F.; Soares, P.B.F.; Versluis, A. & Soares, C.J. Dental trauma splints for the mixed dentition – A finite element analysis of splint material, splint extension, missing teeth, and PDL representation. *Dental Traumatology*, 2022, **38**, 495– 504.
doi: 10.1111/edt.12777

CONTRIBUTORS

Dr Aishwarya Pandey completed her Bachelor of Dental Surgery (BDS) and Master of Dental Surgery (MDS) in Periodontics from the Institute of Medical Sciences, Banaras Hindu University (BHU), Varanasi. She has served as a Research Associate at the Centre of Advanced Maxillofacial Research, DRDO-INMAS, Delhi (2023-2024). Currently, she is working as a Senior Resident at the Institute of Medical Sciences, BHU, Varanasi. His contributions to the manuscript include: conceptualisation, literature search, data collection and analysis, manuscript preparation, and editing.

Lt Col Parvinder Sharma is a Prosthodontist and currently in-charge of the Centre of Advanced Maxillofacial Research at DRDO-INMAS, Delhi. His research interests are application of 3-D printing in dentistry, and development of indigenous dental biomaterials.
His contributions to the manuscript include: conceptualisation and design, supervision, as well as critical revision and editing.

Dr Gayathrie B. earned her Bachelor of Dental Surgery (BDS) from Meenakshi Ammal Dental College, Chennai and Master of Dental Surgery (MDS) in Prosthodontics from SRM Institute, Chennai. She is presently working as a Senior Research Fellow at the Centre of Advanced Maxillofacial Research, DRDO-INMAS, Delhi.
Her contributions to the manuscript include: literature search and data collection, as well as review and editing.