JOURNAL OF TECHNIQUES

Journal homepage: http://journal.mtu.edu.iq

REVIEW ARTICLE - MEDICAL TECHNIQUES

Finite Element Modelling Based Studies for Dental Implants: Systematic Review

Shahad Mohammed Shakir[1*], Saja Ali Muhsin[1], Raad Al Marza[2]

[1] College of Health & Medical Technology - Baghdad, Middle Technical University, Baghdad, Iraq

[2] University of Sheffield, School of Clinical Dentistry, Sheffield, UK

[*] Corresponding author E-mail: shahadmohammed884@gmail.com

Article Info.

Article history:

Received
28 July 2022

Accepted
09 September 2022

Publishing
15 November 2022

Abstract

Finite element analysis (FEA) has been used to evaluate dental implant designs, superstructure structure; material, and surrounding bone stability. According to PubMed, and Google scholar, much FEA research on dental implants was published between 1988 and 2022.
Finite element analysis is an advanced technology used to examine implant-abutment links, dental implant design, and implant screw architecture to verify its usability and dependability in the field of dental implantology.
The purpose of this FEA literature study was to go over the concept applications of the finite element method (FEM) in dental implantology.
Many health-related problems, notably in dental implantology, can be swiftly and simply solved utilizing finite element methods, which combine strength, stress, material science, and architecture. Finite element methods not only give speedy and reliable data on the patients under investigation, but they also act as essential guidance for many clinical trials. To have a better understanding of the finite element modelling method.
For many years, the FEA approach has been employed in medicine and is frequently used in the study of dental implantology. It is a useful tool for permitting endless replication of studies that cannot be duplicated clinically in one-to-one circumstances in various settings.

Publisher : Middle Technical University

Keywords: Finite element analysis; Dental Implant; Stress analysis; Stress distribution; Implant screw.

1. Introduction

The finite element modelling (FEM) was invented at the ideal time as computer capabilities and human talent were demanding quicker solutions to cope with the increased need of economical product development[1]. The finite element approach, usually created in the aerospace sector in 1956, is a more contemporary technique for stress analysis (FEM). This technology was once widely used in aerospace engineering, but it gradually gained popularity in dentistry due to its adaptability to model any complex geometries and deliver immediate results [2]. To replace photo elasticity tests, it was originally applied in dentistry in the 1970s [3].
Verification is defined as "the process of determining that a computational model accurately represents the underlying mathematical model and its solution," while validation is defined as "the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model" by the American Society of Mechanical Engineers Committee on Verification and Validation in Computational Solid Mechanics. Simply put, the process of "solving the equations right" is known as verification, whereas the process of "solving the right equations" is known as validation. To determine the modelling error, computational predictions and experimental data are compared during the validation phase. Rather than testing particular scientific theories, the main aim of these "experiments" is to provide data for comparison with model predictions [4, 5].
Recent developments in the biomechanical area have increased the use of validated FEA research. FEA validations can be categorized into two categories: (1) direct validation, which includes experiments on the quantities of interest (ranging from basic material characterizations to hierarchical system analysis, such as model experiments and in vitro experiments), and (2) indirect validation, which includes the use of literature or the findings of prior clinical studies. Given its iffy experimental quality, sources of error, and high level of unpredictability, indirect validation is unquestionably less preferred than direct validation. The majority of FEA investigations of force distribution, however, might not be linked to any specific biological consequence, making indirect validation in FEA inevitable. As a result, it is challenging to produce outcome data for comparison with experimental data [5].
This study explores the fundamental concept and applications of (FEM) in dental implants; understanding the biomechanics and biomimetic approaches of the tooth in restorative dentistry makes the use of the finite element (FE) method, which can reveal otherwise inaccessible stress distributions inside the tooth-restoration complex. For more dental and oral health science applications, non-linear FEM solutions should be enhanced. Stress analysis of dental structures and associated tissues has become relatively widespread in recent years to determine the stresses induced in these structures and assess the biomechanical qualities of materials utilized for therapeutic purposes in dentistry [6]. Due to significant costs, ethical concerns, and technical problems, the planned investigations in dentistry will not be carried out.

Nomenclatures			
FEA	Finite element analysis	TISP	Tooth-implant-supported prosthesis
FEM	Finite element modelling	NR1	Non-rigid connector one-piece abutment
3D FE	three-dimensional finite element	NR2	Non-rigid connector two-piece abutments
VM	Von Mises	NR3	Non-rigid connector three-piece abutments
RP	Branemark regular platform	R1	Rigid connector one-piece abutment
GPa	Gega pascal	R2	Rigid connector two-piece abutment
RFA	Resonance frequency analysis	R3	Rigid connector three-piece abutment
Y-PSZ	Yttrium-partially stabilized zirconia	PEEK	Polyetheretherketone
SED	Strain energy density	CRPD	Conventional removable partial denture
MDI	Mini Dental Implants	FPD	Fixed partial denture
Co-Cr	Cobalt-Chromium	Zr-S	Monotype Zirconia Implant
Ti-Zr	Titanium- Zirconia	YTZP	Yttria-stabilized tetragonal zirconia polycrystal

Therefore, the research on the stress analysis may require to be conducted on a living tissue model. To determine the object's stress concentration point and make it stronger with extreme durability against any potential forces, various stress analysis techniques are applied [7].

2. Eligibility Criteria, Information Sources, and Search Strategy

A literature survey was conducted from 1988 to May 2022 to identify all aspects of the studies that examined the role of FEM in dental implants. PubMed, Google Scholar, and Science Direct.com were searched using keywords FEM and Dental Implants; FEA; Mono implant screw stress analysis; Single screw with their respective abbreviations according to the search engine used.
Study selection
over 200 results were found; after review, full-text articles published only in the English language and then reviewed for title or summary were included in the review. Research articles with emphasis on 3D FEM and stress evaluation; stress distribution on bone and implant and effect on jaw bones were considered for review; this review article will concentrate on studies over the past 20 years to determine its applicability and dependability in the field of dental implantology.

2.1. The methods of stress analysis

Two methods of non-destructive experimental stress and strain analysis are presented here.

2.1.1. Photo elastic stress analysis

This is a method of Photo elastic stress analysis meter which is responsible for describing the material's optical properties variations under mechanical distortion and it's usually used to determine the material's stress distribution; the analysis gives instant qualitative details about the general distribution of stress, the positions of stress concentrations and areas of low stress. Thus, it can be utilized in modifying designs to reduce or disperse stress concentrations or to remove overflowing material from low-stress areas, thus obtaining a lowering in weight and material costs [8].

2.1.2. Finite Element Analysis

It is a computerized process of speculating how a product will react to real-world forces, heat, fluid flow and other physical effects, and as a result, FEA reveals whether a product will break, wear out, or carry out the way it was designed; it's a cutting-edge scientific technique that provides mathematical equations with numerical solutions to represent a physical model. In the 1950s, the FE analysis was first discovered to resolve mechanical problems in airplanes and space factories and industries, and it is now widely used in a variety of domains, including static analysis, fluid mechanics, electromagnetic analysis and analysis of heat transfer [6, 8].
These days, both medicine and dentistry use it extensively, as well as most engineering fields. The research of Ledley and Huang in 1968 is the earliest documented study of FE analysis in dentistry. This experiment used a numerical tooth model subjected to forces in various directions to investigate the produced stresses in the surrounding bone tissues [9].
The work of FE analysis is all about breaking down the structure of the FE model into finite elements, and then its force response will be analyzed numerically. This technique can be used to undertake a one-, two-, or three-dimensional study of a structure. The computer-created models calculate the distributions of stress, intensities along with shape changes produced as a result of the force applied in the area [2, 10]. Even though the problem is complex, it is broken into simple sub-units to be handled using the FEA method, and every structure is resolved independently by being divided into several discrete pieces known as finite elements. The concept of moving from piece to piece is used to solve the problem, which is made easier by the use of numerous tiny, interconnected pieces [11, 12].
Finite element analysis has several advantages. The FEA has made the modelling of complex solids possible and the stress distributions can be thoroughly examined. Moreover, it allows for the examination of constructions built of various materials. Physical definitions of materials can be used to obtain real definitions. It is possible to replicate craniofacial and dental structures. It is reproducible and repeatable. It takes less time to complete than experimental research [13, 14].
Finite element analysis, despite its many benefits, has certain drawbacks. Computers and analysis software is expensive. The requirement for well-trained and experienced individuals who are familiar with the program is under consideration. The responsibility to keep software programs up to date as technology advances. For the correctness of the research, a detail-oriented transfer of the real system to the virtual environment is required. The modelled material's properties, such as isotropy, homogeneity, and linear elasticity, do not entirely mirror reality [14, 17].

3. Stages of Finite Element Analysis

There are three initial steps to establishing finite element analysis.

3.1. Pre-Processing

According to Erden and Yayla, 2021, the following processes were stated "One of the most important factors to consider when obtaining a solution using the FE analysis method is the mesh size. The mesh size used is important so that when the model is analyzed, it converges to the correct result. As the mesh size decreases, the number of elements used will increase as the model is divided into smaller pieces. When the number of mesh is increased, the accuracy of the analysis results also increases" [15]. Networking is considered as the analysis process foundation which employs the FEA method. The coordinated elements are linked together by the nodes which were created throughout the networking procedure [8]. The nodes represent the sites where the elements join one another, and the mesh is the overall structure. As they are generated in the network model, adjacent elements do not overlap and do not have any gap between them. The elements should have the simplest feasible structure. In one dimension, lines are favoured, in two-dimensional structures, triangles or parallel edges come next, then four-faced, five-faced, and six-faced structures in three-dimensional structures. Nodes, lines, and planes partition one-dimensional objects into finite elements. Nodes connect all of the elements that make up the object. "A network of finite elements linked by nodes will so substitute the body." The term "body" refers to the problem's ongoing context or area. Nodes, on the other hand, function as nut and bolt connectors, joining adjacent finite pieces at their ends. When nodes are eliminated, physical continuity between adjoining finite elements is lost because the elements separate [8]. The method of analysis should define each element's matrices representing the object. Then the matrices of the element were added together to generate the segmented object's general matrix. The balance of forces and displacement continuity In every node of the body's finite element model [8] is presented in this collection. Following that, the model is subjected to boundary conditions. Stresses and displacements are boundary conditions. It depicts where the object's fixation is located as well as the force application. The condition of the object is what determines it. The application of boundary conditions is defined by the region of the studied object [10].

3.2. Solution

After the forces have been applied, the equations generated are resolved between the finite number of elements [16].

3.3. Evaluation

The results step's evaluation (post-processing) is where the equations' solutions are visualized using charts, figures, and colour visuals [17].

4. Finite Element Analysis: Basic Mechanical Concepts

4.1. Force

Force is the property that allows objects to change their speed and shape. External force refers to the effect of other items on the object under investigation, whereas internal force refers to the effect and reaction force between the object's many sections. The force represents a scalar quantity with vectorial qualities such as violence, magnitude and direction. Compression tension, tensile stress, and shear forces are the three types of forces that exist; The relationships between the loading force and stress on the abutment and alveolar bone were evaluated in a study by Kang et al.; the appropriate range of stress on the abutment suitable for alveolar bone remodeling is 1.5 MPa -8.66 MPa, and the appropriate range of the loading force is 6 N-86 N [18].

4.2. Stress

An object's reaction to a force applied in the opposite unit area to its centre is called tension. When the tension is applied the body's interior structure can be affected more than the molecular structure does. The force per unit surface in a structure is called tension. Pascals, kg/cm^2, or N/m^2 are the units used [19, 20].

$$\text{Stress (S)} = F/A$$

Where
S: Stress (Psi or Ibs of force per $in.^2$)
F: Applied force (Ibs of force)
A: Cross-sectional area ($in.^2$)

Depending on the applied force, there are three forms of stress (stress):

1. Tension perpendicular to an object's surface that drives its molecules to separate, is brought about by two forces that are both acting in the same direction. Tensile stress occurs when a force tries to stretch or stretch items, causing distortion.
2. Compressive stress occurs when a force attempts to squeeze or shorten an object.
3. Is a kind of tension produced by the action of two forces in opposite directions and at different levels, forcing the body's particles to slide parallel to the surface and in opposite directions, one on top of the other like layers. The most harmful stresses for materials used in fixings are thought to be cutting (shear) stresses [19, 20]. The maximum computed stress values in preloaded screws after occlusal pressures increased or decreased; these maximum stress values were considerably below the yield stress of both abutment and prosthetic screws of the two implant systems examined. The findings show that the three implant-to-abutment joint systems evaluated are unlikely to fail under simulated occlusal forces.[21].

4.3. Strain

Strain is defined as the proportion of an object's original size to its dimensional change as a result of a force being applied to it. Deformation at any point on the body when it is being loaded is another possibility. The pre-force state illustrates the degree to which the material has changed after force application. In most cases, strain is reported as a percentage (percent). The force keeping the construction blocks together will outweigh the applied force if the latter is larger than the object's maximum tension, leading to rupture or breaking, Strain equals Shape change/original length [22]. Tada et al., study confirmed the significance of bone quality in implant long-term prognosis. Implant length and type can significantly affect bone strain, especially in low-density bone models; in particular, under an axial load, maximal equivalent strain in cancellous bone was lower with the screw-type implant than with the cylinder-type implant in low-density bone models. It was also lower with longer implants than with shorter implants [23].

4.4. Principal stress

The stress has no impact on any plane, and the principal stresses are those in which the current stresses only work perpendicularly. Maximum, middle, and minimal primary stress are the three types of prime stress. Tensile stresses are represented by maximum principal stresses (Pmax), which are positive. The maximum stress (compression) stresses are expressed by the minimum principal stresses (Pmin), which are negative [24, 25].

4.5. Von-Mises stress

The commencement of deformation is determined by Von Mises stress, which is employed for retractable materials. It's used to find out about the material's stress concentrations and distributions. Combining two- or three-dimensional stresses yields a calculation of the tensile strength of single-direction loaded material. Von Mises (VM) is also utilized in stress and fracture resistance calculations [26].

4.6. Module of Elasticity (Young Module)

It measures how resistant a thing is to deform. The modulus of elasticity increases as the stiffness of the items increases. Pascal (N/cm^2) is its unit. It's the stress-strain ratio applied at any point of the object. The "Young module" term is referred commonly to as the elasticity module or elastic coefficient [27, 28]. Piotrowski et al., investigated a dental implant with four different Young's moduli and two forms; they discover that the elasticity modulus has a significant impact on load transmission (stress) between the implant and the bone contact. The stress jump at the interface between the cortical bone and implant was considerably minimized for each evaluated configuration by employing a low Young's modulus implant. As a result, the micro-motions at the cortical bone-implant contact are minimized [29].

4.7. Poisson Ratio

The application of the pulling force leads to a decrease in the length of the section, an increase in the item's length, as well an increase in the section's thickness. The Poisson ratio is the ratio of changing the body's dimension from the lateral dimensional change to its axial dimensional change when it is subjected to tensile and compressive stresses (v). Depending on the material, the Poisson ratio changes. There isn't a unit for this rate. The material continues to be reduced in cross-section until it breaks. The Poisson ratio is larger for softer materials, which show more cross-section reduction during drawing [6].

4.8. Hook's law

It is a linear relationship between both stresses and voltages. It approximates the behaviour of items as long as they don't exceed predefined stress limitations. The tension-stress curve is used to determine the object's deformation throughout force application. The force coefficient is determined by this curve's straight slope; it displays the item's rigidity (k). It asserts that the flexible material has a low elasticity coefficient while the hard material has a high one. ($F=-km$) is the formula. The formula's negative sign shows that the force always acts in the displacement's direction [30].

4.9. Deformation

After an object is applied, it will distort either temporarily (elastic deformation) or permanently (plastic deformation). Elongation and shortening are two types of deformation brought on by tensile stresses, while compressive forces cause compression [24]. The results of the tests showed that the performance of the Morse taper implant was greater than the performance of the external hexagonal implant when both were tested cyclically in samples of different densities; this increased the long-term stability of these implants in the clinical application; It has been demonstrated that the diameter, length, density, and kind of implant-abutment interface are design elements that influence implant behaviour. The deformation and stress findings from the penetration FEM model follow the same pattern as the analytical and FEM indentation results. [31].

4.10. Resistance to Fatigue (Yield Stress)

Plastic deformation occurs when a material's maximum resistance is exceeded. Fatigue resistance refers to the line separating elastic and plastic deformation. The object is inelastic when it's reaching to a point higher than the yield point, however, it can be restored after stress removal [22]. Gil et al., when compared to the internal hexagon interfaces, the fatigue behaviour of the external hexagon interface produced better results. The size of the resistant part is what gives the external connection its long fatigue life. The fundamental reason for the changes in mechanical properties is that this fact leads to a better load distribution of the load [32].

4.11. Flexion

The force exerted on the object causes it to rotate around its axis. On the bent surfaces of the object, mutual compression and tension forces exist [22]. El-Anwar and El-Zawahry investigated the distribution of stress in adjacent bones were investigated for a gradual increase in implant diameter and length; utilizing the finite element method to determine design curves, several loadings were applied to each design, including bending of 20 N and torque of 2 Nm. The outcome suggests Von Mises stresses, which decrease the ratio of side to cross-sectional area, were seen on spongy bone as a result of bending loads. Finally, torsion stress results in cortical and spongy bones. While implants with smaller diameters may perform better as their length is increased, implants with bigger diameters are more stable (side area) [33].

4.12. Isotropic Material

Regardless of their coordinate system, they are acting as the same materials. Their similar mechanical properties and elastic qualities response are in all directions when force is applied in various directions [22].

4.13. Orthotropic Material

They display a variety of mechanical properties when the force's application is accomplished in different directions. It's an elastic modulus change depending on the direction of the force. As an illustration, bone tissue [22].

4.14. Element (Element)

The term "element" refers to the basic geometric shapes utilized in finite element analysis. Depending on their size and shape, workers are classified as rotating elements, 1D, 2D, or 3D elements, such as triangles, parallel edges, and rectangles [34].

4.15. Node

In finite element analysis, nodes are the linking points where elements are located and whose behaviours have been stated on a computer-built model [24].

5. Finite Element Analysis Researches in Dental Implantology

In the past two decades, many types of research using the finite element analysis technique have been published in the branch of implant dentistry; due to the difficulty of acquiring data in *vivo* study, Table 1.

Table 1 Application of Finite Element Analysis (FEA) in implant dentistry

Author	Study Variables	Conclusion
Holmgren et al.	The investigation of research for 3D FEA was also conducted under a variety of therapeutically relevant conditions, including vertical, horizontal, and 45° oblique 3 loading at the top of the transmucosal implant- abutment.	Compaction stresses were comparatively higher when the implant was tilted, regardless of the loading's site and direction. Testing with an eccentric loading and a 45° loading direction made this tendency more obvious. When the loading of 45° was applied to the inclined model, the compressive stresses were found on the cortical bone next to the inclination direction and tensile stress on the opposite side [35].
Akça et al.	Compared the methodologies of strain gauge analysis and finite element analysis. Both methods produced similar results regarding the amount of stress generated by the applied forces.	Found that the finite element analysis method offered better modelling benefits because it provides for more sensitive and precise results. It is commonly accepted that "isolation" of the amount and direction of forces transferred to each implant and/or prosthetic component is necessary for reliable measurement of strains on an implant-supported prosthesis to make improvements in design and treatment [36].
Geng et al.	Examined a finite element analysis in two dimensions was used to evaluate an osseo-integrated stepped screw dental implant (FEA). A digital cross-section of the human mandible's posterior side was used for implant modelling; the mandible's cross-section was taken from a patient data collection created by computed tomography (CT). A 15mm Branemark implant with a regular platform (RP), its length and diameter of the neck were as same as a control. Among the therapeutically relevant situations under which the experiment was done, the transmucosal abutment loading was in three	The study found that the elastic moduli of the cortical bone and an oblique load are crucial factors in implant design optimization. The stepped screw implant produces a more uniform stress distribution than the cylindrical screw implant; It is believed that both an improved stress distribution and reduced maximum stress in the trabecular bone are the results of the lower stiffness of the stepped screw implant. The study also showed that, unlike the Branemark implant, which calls for a minimum cortical bone modulus of 13.4GPa, the stepped screw implant is appropriate for cortical bone moduli ranging from 10 to 13.4GPa [37].

	orientations: horizontal, vertical and 45° obliquely.	
Lin et al.	A study assessed the influence of bone quality and implant length on the biomechanical characteristics of dental implants and alveolar bone using non-linear finite element analysis. In comparison to implant lengths and bone quality, loading conditions were determined to have the greatest impact on the biomechanical features of alveolar bone and implant systems.	The results of the simulation suggested that more research is necessary to fully understand how occlusal modification affects the loading directions, which could have a long-term influence on the viability of the implant system [38].
Lin et al.	The study aimed to understand how much the implant system and its position with bone categorization, and loading situation influence the biomechanical outcomes of implant-supported restoration with single-unit. The mechanical reactions of an implant inserted in the posterior region of the maxilla were simulated using nonlinear finite element analysis.	Implants positioned parallel to the loading axis show better stress/strain distribution. It is advised to put implants and make certain occlusal adjustments to lessen lateral tension. As a force-transmission mechanism, an implant with a tapering interference fits connection outperformed other designs [39].
Quaresma et al.	An investigation into the effects of two dental implant systems which are commercially available on the prosthesis stress distribution, abutments, implants, and supporting alveolar bone when subjected to simulated virtual occlusal forces uses FEA	This results in the stepped type of the cylindrical implant's connection to the internal, hexagonal abutment with screw retention, while, the abutment complex is subjected to less stress. In contrast, the alveolar bone and prosthesis are subjected to lesser strain from the conical implant when it is connected to a solid, internal conical abutment and higher stress from the abutment [40].
Kong et al.	Investigated the impact of implant diameter and length on the jaw's Von Mises maximum stress and maximum implant-abutment displacement under instant loading models was examined using a non-linear FE approach.	Under axial loads, the implant was found to be more crucial in strengthening its stability under buccolingual loads. The minimal stress and displacement were obtained with more than 4 and 11mm, respectively. These dimensions were thus the optimal ones from a biomechanical standpoint for implants with immediate loading [41].
Silva et al.	Studied the rehabilitation of maxillary edentulous area with six dental implants using the all-on-four technique in with FEA approach.	They have found that the all-on-four technique had greater stress, especially if there was a cantilever, and that the cantilever sections had more stress. Both models had similar stress distribution and placement characteristics. The Von Mises maximum stress values were decreased by implant inclusion. The stress was substantially exacerbated by the cantilever [42]
Winter et al.	Investigated the effect of several factors, including the length of the implant, cortical bone thickness, dimension and damping factor which are described as bone quality, bone loss, and the transducer fixation effectiveness, in which the Resonance Frequency Analysis (RFA) and damping factor capacity measurements can be affected.	Increases in osseointegration were found to increase implant stability for all parameters. The dental implant’s length and cortical bone thickness were positively connected with implant stability, with values somewhat convergent at higher levels of osseointegration. Changes in bone's damping factor showed no discernible impact. When osseointegration levels were higher, there was a small convergence in the values for the relationship between implant stability and bone loss. Implant stability values were affected nonlinearly by the linear changes that occur in the length of the implant and also in bone resorption [43].
El-Anwar and El-Zawahry,	Examined 25 distinct designs of a dental implant with incremental increases in both length and diameter were studied in 3D utilizing FEA to derive simplified numerical design equation for better implant behavioural understanding	The results of the analysis demonstrated that the enhanced stress distribution on different bone areas (cortical and spongy) is produced by increasing implant diameter and length. This investigation led to the creation of approximate design equations and curves [33].
Yeung et al.	Investigated an osseo-integrated implant that was inserted with the surrounding bone that made of commercially purified titanium (Ti) or yttrium-partially stabilized zirconia (YPSZ) combined with various superstructures made of either gold alloy or YPSZ in the posterior region of the maxilla is the subject of a study that examined and contrasted the stresses of the bone-implant interface in two different situations. Moreover, this study analyzed the	In conclusion, the two interfaces, the maximum Von Mises and compressive stresses showed a lower result in the implants made of zirconia groups compared to the implants made of titanium groups. Both types of implants' apical regions and the palatal side of the platform showed the same stress distribution pattern in the cancellous bone [44].

	stresses of the bone-implant interface in two different situations. Then compared it in anisotropic 3D finite element models (FEMs) of an osseo-integrated implant. It is constructed using either commercially purified titanium (Ti) or yttrium-partially stabilized zirconia (YPSZ). These are combined with different superstructures fabricated in the posterior region of the maxilla using either gold alloy or YPSZ.	
Barão et al.	Studied the effect of over dentures with implant retention and implant-supported full-arch prostheses with various designs on the edentulous mandible stress distribution was compared using a finite element analysis. Four distinct models of the mandible were constructed. For all groups	The trabecular bone had a low-stress concentration compared to the cortical bone. The stresses in the implant surrounding tissues, mucosal tissue, and the components of implant/prosthetic structure were minimized by using implant-supported dentures and non-splinted implants retained removable dentures to restore the mandibular dentition [45].
Wen et al.	Investigated three different zygomatic implant techniques, including the severely atrophic maxilla.	The externalized technique of restoring the lateral incisor with only one implant appeared to be the most appropriate rehabilitation method for the severely atrophied maxillary edentulous arch. All three techniques of inserting a zygomatic implant somehow showed in a more or less force-consistent transference and could therefore rehabilitate the edentulous maxillary arch [46].
Rezende et al.	The single implant stress distribution was also examined in a (FE) model as an alternative method to an in-vitro model. The Brnemark implant, a multiunit set of (5mm) height abutment, a screw-retained metal-ceramic crown, and a polyurethan material to simulate bone made up the in vitro model. Strain gauges were used to measure the mesial and distal deformations in the area surrounding the implant following the application of an axial load of 300N in the centre of the crown’s occlusal aspect. Micro CT was used, and the strains were recorded in the vicinity of the implant. The stress distribution in various system components was assessed using the FE model.	The cortical bone experienced strains of 5.83 and 40MPa, the implant experienced stresses of 55 and 1200MPa, and the abutment screw experienced stresses of 80 and 470MPa, respectively, due to axial and oblique loads. The deformation around a single implant was successfully assessed using the FE approach. A larger concentration of stress is caused by oblique loads [47].
Moraes et al.	Using a 3D FEM, the stress distribution of prosthetic implants screwed to clinical crowns with different heights was inspected and studied. The crown/implant system's weakest link is the retention screw of the implant-supported dental prosthesis. Another significant aspect that may raise the lever arm is crown height. As a result, the purpose of this study was to evaluate the stress distribution of prosthetic implants screwed to clinical crowns with different heights using the three-dimensional finite element approach. The creation of three models was done using crowns and implants (3.75mm and 10mm) (heights of 10, 12.5 and 15mm).	The findings were visualized by using the stress maps of Von Mises with increasing the crown heights. Stress levels in the oblique load were higher in the screw construction. Comparing the oblique loading to the axial loading, a greater stress concentration was obtained. The screw on which the stress distribution occurred was harmed by the increase in the crown, especially in oblique loading, it has been determined [48].
Bhering et al.	Studied the prosthetic rehabilitation of atrophic maxilla, researchers compared all-on-four and all-on-six therapy ideas. A full-arch fixed restoration was used to restore an edentulous maxillary arch with a slight sinus pneumatization in a prototype that served as the foundation of the 3D FEA. The all-on-four approach involved placing four typical dental implants, and the concept of all-on-six involved placing four typical dental implants and two short ones. Six groups of three different framework materials titanium (Ti), zirconia (Zr) and (Co-Cr) examined the displacement levels on the cortical bone	Ti showed the highest stress values since The absence of distal support for the framework and Ti low elasticity modulus; It had been concluded that the design of all-on-six and stiffer framework materials demonstrated the greatest biomechanical behaviour. For both treatment ideas, the stress levels did not, however, go above the bone resistance limitations [49].

Gosavi et al.	An experiment on the strains at the bone-implant interface for various types of dental implants is being investigated and monitored. Photo-elastic analysis of the stress was performed on four implants with different lengths and diameters which are commercially available, using the finite element analysis to verify the experimental results.	The cancellous bone had the most stress in the neck region for all models. Both the FEA and experimental photoelasticity produced the same results. Stress can be reduced by increasing the implant's diameter [50].
Eom et al.	Compared the treatment of implant-supported prosthetics to other prosthodontics treatment alternatives. Based on computed tomographic information from a patient, partial maxilla, teeth, and prosthetics finite element models were created. The model was generated with the teeth, examined crowns, and RPDs. The produced components were used to create four 3D finite element models of the partial maxilla: tooth-supported RPD (TB), implant-supported RPD (IB), tooth-tissue-supported RPD (TT), and implant-tissue-supported RPD (IT). The crowns and denture teeth received an oblique 300N loading.	The denture's abutment and implant system's Von Mises stress, as well as displacement, were found. The stress distribution pattern of the implant-assisted distal extension removable partial denture (IARPD) was distinct from that of the RPDs of natural teeth, according to a finite element study, and it was also different from that of the RPDs supported by implant tissue. More thought must be given to the RPD design and the quantity or placement of the implants when they are used as RPD abutments [51].
de Souza et al.	In the posterior maxillary implant-supported prostheses, the cantilever extension is compared to different treatment choices in this study.	Using the central pontic revealed more favourable distributions of both stress and strain in the examined components of the two prostheses supported by implants. The usage of the cantilever revealed negative biomechanical behaviour, particularly for the distal cantilever. Lower stress and strain values were seen on the examined structures when three implants were used. The cantilever-equipped prostheses, which were supported by two implants, displayed undesirable biomechanical behaviour in the examined structures, particularly for the distal cantilever [52].
Bramanti et al.	A study established emphasizes the distribution of stress over several conventional crown restorations using Computer-Aided Design (CAD) tools with a 3D computer-generated model. By using engineering specialized analyses systems like FEM and Von Mises investigations, the strength over simulated mandibular first premolar crowns restorations fabricated from chrome cobalt (Cr-Co) alloy, golden alloy (Au), dental resin, and zirconia (Zr) materials. Prosthodontic crown models have been made, and stresses resembling the chewing force have been simulated.	The forces applied to the 3D models were in axial and oblique directions, and both ensured the desired outcomes during a simulated masticatory cycle. While zirconia and metal alloys recorded high values for fracture, dental resin showed a low value. For the teeth they wish to replace and restore, doctors should select the best prosthetic option. Both artificial dental crowns are successful over the long run if they are used following the manufacturer's limitations and recommendations [53].
Aumnakmanee et al.	The research effort focuses on a three-dimensional finite element static simulation for dental implants. Four different thread designs have various effects. The dental implant models are subjected to compressive forces of 60 to 200N and shear forces of 20N with a forced angle of 60°.	The findings of this study help to provide a better understanding of the stress distribution characteristics and this can be used for a better design of threaded dental implants [11].
Wazeh et al.	The effect of dental implant threading characteristics and the choice of material are also thoroughly examined in the bone of the mandible under crown restoration with two different materials which are Translucent Zirconia and Porcelain fused to metal. The analysis and comparison of the findings of 24 case studies were conducted.	The Von Mises stress in micro thread implant was reduced by 50-70% than the conventional thread one. Higher Von Mises values seem to be produced on the implant body by a 50N oblique loading and then by a 100N vertical loading, which is by a factor of 4 to 5. When the tested crown material was switched, there was no effect on the cortical bone or very little of one. In contrast to reinforced PEKK (poly-ether-ketone-ketone) or PEEK, titanium implants can also significantly lessen the cortical bone, Von Mises stress by a factor of 50 to 100. Implants made of reinforced PEKK and PEEK can be considered alternatives to titanium ones. A crown made of zirconia distributes applied loads more evenly than one made of porcelain bonded to metal. Despite of implant's

		material, a micro threaded implant behaves better than a standard implant. The greatest alternative for the patient's bone may be the zirconia crown placed over the titanium implant [54].
Liu et al.	The effect of implant orientations and the loading times on stress distributions in the bone tissue surrounding the implant was studied using immediate-loading and delayed-loading models. Different posterior implant inclination angles (0°, 15°, 30°, and 45°) were incorporated into four 3D FEA models. The stress distributions were measured in the bone tissue surrounding the implant before and after the osseointegration process. The bone maximum principal stress of tensile, mean tensile, and compressive was also computed.	Peak main stresses were found in the bone area around the left-tilted implant in all of the models. The 0° model had the highest maximum and minimum values for both models with immediate loading and delayed loading. Furthermore, the maximum and minimum values for the 0° and 15° models were greater. The greatest micromotion was visible in the 0°models. Compared to the 0° and 15° models, the 30° and 45° models performed better in terms of the observed stress distribution [55].
Wu et al.	Using in-*vitro* tests of strain gauge and 3D FE computations were examined in the terms of the biomechanical implications of an implant design and placement of the loading on the implant and the bone surrounding it, an all-on-four procedure using four implants osseointegrated in the bone is superior. Based on the all-on-four treatment idea, in *vitro* and 3D FE models were created by inserting titanium frameworks, NobelSpeedy, and NobelActive implants into an edentulous jawbone. Three different types of loadings were used: one at the central incisor region and the other at the molar regions with the denture cantilever, and one without (loading position 3). Rosette strain gauges were used to record the main bone stresses for the in vitro testing, using 3D FE models, the peak Von-Mises stresses in the inserted implant and the cortical bone surrounding it were examined and analysed statistically using Wilcoxon's rank-sum.	The surrounding bone's peak strain and stress (in the 3D FE study) were typically about 36-62% and (in-vitro test) about 47-57% significant values for loading position 3 compared to 1 and 2 loading positions. The bone stresses and strains between those 2 types of the dental implant were comparable. Changing the implant design for an all-on-four procedure using 4 dental implants which are osseo-integrated with surrounding bone doesn't seem to have an impact on the entire procedure's biomechanical performance, particularly in terms of the surrounding bone on which stresses and strains are applied [56].
de Melo et al.	The study examined a 2.9mm narrow implant used in an all-on-4 implant system to support the fixed restorations at the posterior region of the maxilla. This is compared to the 3.5mm wide implants in terms of stress implants. With 2 anterior implants in parallel position and perpendicularly inserted to the crest of the bone and two 30°angled posterior implants, two narrow-diameter implants of 3.5/11.5mm (Unitite Prime) and 2.9/11.5mm (Unitite Slim) were used in simulated all-on-4 therapy scenarios to compare how well they performed with a masticatory force under both axial loading and oblique loading of 100N. To represent maxillary implant-supported full prostheses, a computed tomography-generated model of the edentulous maxilla was merged with a parametric CAD model of the prosthesis. The Mohr-Coulomb criterion and the peri-implant bone were evaluated. The Von Mises criterion and the Rankine criterion were used to evaluating abutment teeth, dental implants, and screws, respectively.	It was believed that the osseointegration would be fully developed. In comparison to the 2.9mm model, the 3.5mm model displayed higher values of the axial load for the bone surrounding implants, dental implants, and abutment. In terms of oblique load, the 3.5mm model had greater values for right-sided bone surrounding implants, dental implants, abutment, and frames than the 2.9mm model did. For the axial loading and 4% for the oblique loading, the risk of the bone surrounding implants was lower in the 3.5mm type by 16% and 4%, respectively. The 3.5mm type showed higher peak stress on implants and abutments than the 2.9mm model despite having a decreased loss risk of the bone surrounding implants [57].
Valera-Jiménez et al.	Developed a new design of the dental implant for the desired stress distribution in the bone surrounding area using FEA. A 3D replica of a real maxilla was built using images taken by the CT, and numerous implants; Narrow-diameter implants, regular-diameter implants, wide-diameter implants (NDIs, RDIs, and WDIs) and restorations were made using the CAD software.	The mechanical advantages of the concept of splinting were shown, most significantly in that the bone surrounding implant overloaded volume NDIs splinted with the 3-unit bridges were smaller than that around the non-splinted implants with a larger diameter RDIs however, splinted NDIs supporting all-on-four prostheses were linked to the study's highest danger of overloading. This is due to placing weight on the cantilevered molar

	Biting forces in correspondence to 3 different rehabilitation procedures were modelled on the prostheses, including single-unit implant-supported restoration, 3-unit bridges, and all-on-four therapy. A cylinder of 0.1mm diameter ringed the bone surrounding each implant to measure the bone overloading inside volumes. These calculations were used to determine how much stress was distributed around the implants.	enhancing the compressive stress that was already present around the implant's angled surface [58].
Tribst et al.	FEA based on the framework design was used for mechanical behaviours evaluation of 2 different maxillary arch prosthetic reconstructions. Building full-arch implant-supported dental restoration required modelling software. Models were created for both a milling experimental prosthesis and a cast regular framework. The bone geometries, dental implants, restoration, and abutment have all been modelled. A simulation was created for each isotropic and homogeneous material's mechanical characteristics and coefficient of friction. The prosthesis exterior surface was subjected to a 100N load at a 30° angle, and examinations of the results were performed in terms of Von Mises stress, displacements and micro-strains.	Except for the screw of the abutment, which displayed an increase of the stress of about 19.01%, the experimental design revealed a decrease in prosthesis stresses, the strain of the bone and displacement in the metallic. Comparing the experimental design to the conventional design, the experimental design revealed values of higher stress on the prosthetic framework (29.65MPa) between the implants placed anteriorly (13.27MPa in the same region). There is a different design that has a stronger framework and less concentrated stress. In the design and analysis of full-arch implant-supported dental restorations with constrained vertical/occlusal dimensions, this study is a significant milestone [59].
Tonin et al.	The FEA was applied also to assess the torque of the abutment screw that affected the creation of micro-gaps at the interface of implant-to-abutment of a conical bone connection when it was loaded obliquely. This is significant because peri-implantitis is assumed to be caused by germs entering the internal implant space through the abutment/micro-gaps. implants. The conical implant-abutment connection was subjected to 3D FE studies using 20Ncm and 30Ncm screw torques. The prosthesis mounted on the implant received oblique stresses between 10N and 280N. To evaluate the required load to internal implant space, the maximum stress of Von Mises in the abutment's screw and the method to create the micro-gaps were reported	Under oblique loading, the abutment screw stresses were only mildly sensitive to the screw torque. In the absence of an external load, the screw residual stress with a 30Ncm torque was higher by about 35% than that with a torque of 20Ncm. With increasing the load of both torque values, the shrinkage occurs at the contact area of the implant-to-abutment interface. For screws with a torque of 20Ncm and 30Ncm, respectively, the internal implant space bridging requires critical loads of 160N and 220N. The largest gap measured roughly 470μm when all the loads were present. The development of micro-gaps at the interface of implant-to-abutment can be decreased by increasing the screw torque. The abutment screw's mean stress will increase as a result, albeit this could shorten the prosthesis's fatigue life as well [60].
Mohamed et al.	This study used a simulated lateral occlusal scheme to compare the stresses brought on by conventional two-piece (TP) dental implants with those brought on by one-piece (OP) dental implants employed in the All-on-4 concept. 2 FE models were developed for the maxilla, implants, and prosthesis using the concept of all-on-4. Dental implants consist of 2 pieces in the model TP, while, those in the OP model were one piece. 2 scenarios of the loading were applied to each model: the first one simulated a scheme of group function occlusion, and the second one was canine guided occlusion.	. The values of the highest stress were documented for the model TP with the function occlusion group, whereas, the values of the lowest stress were documented for the model OP with the occlusion with a canine guiding. It may be concluded that OP dental implants were with a lower stress value than TP dental implants when they served in the All-on-4 implant-supported restoration of different occlusal/lateral schemes. Lower stress levels are obtained with canine-guided occlusion as compared to the group function occlusal scheme [61].
Bassi-Junior et al.	On prostheses with 3 and 4 implants, the mechanical stress on the metallic prosthetic bar was assessed and compared, along with the dental implant stress. Two 3D human jaw models were constructed. Three dental implants (P3) and 4 dental implants (P4) were provided using different models. Based on the location of the dental implants, prosthetic bars were built for both models. Both of the prosthetic bars' ends were subjected to compression forces using finite element analysis software. Tension and stress dissipation were studied on dental	Prostheses of the P3 and P4 protocol types are effectively force-supported. The dissipation of force along the P3 bar was with higher uniformity than it was along the P4 prosthetic bar. Additionally, compared to P4 implants, P3 implants underwent more stress [62].

	implants and the prosthetic bar imitating force application during mastication. Dental implants and the prosthetic bar underwent analysis without damaging the bar or the implants.	
Zupancic Cepic et al.	Researchers analyzed the biomechanical influences of different prosthetic/implant structures. The orientation of the load on the 3-unit restoration of short dental implants supported the lower arch posterior region using 3D FE models. An atrophied mandible with an absent second premolar, first, and second molars was repaired using two short implants supporting a 3-unit dental bridge or 3 short dental implants (IL= 8mm, 6mm, and 4mm) supporting zirconia prosthesis in splinted or single crown designs. In ABAQUS (Dassault Systèmes, France), under an axial and oblique (30°) load simulations forces of 100N were performed to assess the implant prosthesis stiffness and forces within. The local strain measurements of the implant and prosthesis system and the peri-implant bone Strain Energy Density (SED) were made and compared between the structures.	The overall stiffness of splinted arrangements was almost one and a half times greater than single crowns, with off-axis loading leading to a 39% decrease. As compared to splinted prostheses, single crowns displayed a worse stress distribution. Local stresses under axial load were lower and cannot spread over a wider region compared to the oblique one. In the 2 implant-splinted setups the pressures increased by 25% on each implant as compared to splinted crown restorations on three implants. Un-splinted configurations loaded, boosting the magnitude of the local SED. The size and local stress distribution peaks in the bone surrounding implant regions are greatly impacted by the prosthetic restoration splinting adjacent short implants in the posterior jaw. It is encouraged to use implants to replace every missing tooth when cost and bone availability permit [63].
Huang et al.	This study assessed the tooth-implant-supported prosthesis (TISP) stress distribution following loading in different connectors areas and implant abutments. While, the NR1, NR2 and NR3 were the equivalent tooth-abutment implant systems linked by a non-rigid connector type, R1, R2 and R3 represent the tooth and the one-, two-, and three-piece abutment implant systems, respectively. A 50N vertical occlusal load was applied on the six occlusal regions of the occlusal surface at a straight angle.	In light of the stress distribution’s maximum average, R1 and NR1, therefore, appeared on the fixture of the implant, while, the other 4 models were on the abutment of the implant. However, despite the abutment implant technique, the highest Von Mises stress was produced by the rigid connector compared to the matching of the flexible non-rigid connector type in the cortical bone surrounding the implant. Additionally, compared to the 1-piece and 2-pieces implant systems, the 3-pieces abutment implant system can decrease the Von Mises stress in the cortical bone. It was found that by including a non-rigid connector and a 3-piece abutment system design in tooth-implant-supported prosthesis (TISP); the occlusal load of the implant was distributed and the stress could be regularly presented into a quite strong implant abutment [64]
De Matos et al.	Fixed partial dentures (FPD) with monotype titanium implants, zirconia implants, and two-pieces zirconia implants supporting ceramic abutments were examined in terms of their biomechanical behaviour, stress distributions, and bone microstrain using finite element analysis. A cement-retained implant abutment was modelled with an FPD and implant models with measurements of (4.1×10mm). Three groups of identical geometries were created using these models: Monotype Zirconia Implant, Zirconia Implant, and Titanium Implant and Zirconia Abutment (Ti-Zr) (Zr-S). The first premolar core received an axial load of 300N. As failure criteria, microstrain and Von-Mises stress (MPa) was taken into account.	The three groups' stress was more concentrated in the area next to the FPD connectors. The prosthesis and implant of the Ti-Zr group were more heavily stressed than those of the other groups. However, the titanium implant's lower elastic modulus compared to the zirconia implant resulted in less stress on the abutment and prosthetic screw. The monotype implant system's strain on the bone tissue supporting implant was mostly localized cervically, allowing for a more even distribution of load. Monotype or 2-pieces zirconia implants can be used for FPD treatment. The monotype system, on the other hand, does not have a gap between the implant and abutment, which eliminates stress development in the prosthetic screw and reduces the strain on bone tissue supporting the implant [65].
Tribst et al.	the framework material and the angulation of the distal implants had an impact on the concentration of stress in an all-on-4 full-arched restoration. A full-arch restoration supported by an implant 3D model was provided using numerous angulations of distal implant, cantilever arms (30° with 6mm cantilever, 30° with 10mm cantilever, 45° with 10mm cantilever), framework materials, and cantilever arms [Co-Cr alloy], [YTZP] and [PEEK]). The mesh was created using tetrahedral pieces input	For all configurations with larger stress magnitudes, distal implants with a 45° angle and a 10mm cantilever arm showed the largest stress concentration when the PEEK framework was taken into consideration. With the lowest stress peaks, distal implants with a 6-mm cantilever arm and a 45° angle demonstrated encouraging mechanical responses. If it is possible to shorten the cantilever. Only a distal implant angulation of 45° is beneficial for the All-on-4 idea; otherwise, 30 degrees should be taken into account. YTZP and CoCr concentrated tension inside the

	into computer-aided engineering software from each solid. For each solid, isotropic and homogeneous behaviour was used to give material attributes; they were regarded as bound contacts. A vertical load of 200N was applied on the distal portion of the cantilever arm, and measurements were made of the bone's maximum and minimum main stresses using the Von Misses (VM) method for prosthetic parts and the VM method for bone.	framework as opposed to PEEK, which reduced stress within the prosthetic screw [66].
Jia-Mahasap et al.	A mandibular Kennedy class I RPD supported by a mini dental implant was examined to determine how quantity and location affected the volume average stress and Von Mises stress distribution patterns on the tooth representing abutment, edentulous ridge, mini dental implant, and the bone tissue surrounding. Eight mandibular Kennedy class I finite element models in three dimensions were built, each with a distinct arrangement of micro dental implants. The locations of the first molar, second molar, and second premolar all received little dental implants; therefore, the distally inserted implants reduced strain at the edentulous area by changing mandibular Kennedy class I to a more favourable arch configuration: mandibular Kennedy class III; All models were subjected to a static load of 400N. By using 3D FEA, the Von Mises stress and volumetric average stress were computed.	The lowest volumetric stress average of the tooth representing abutment was calculated using a model with one minimal dental implant in the position of the second molar and two minimal dental implants in the positions of the first molar and second molar. Comparable to the model with two micro dental implants, the model with 3 mini dental implants showed decreased volumetric stress average of tooth representing abutment. However, when three tiny dental implants were used, followed by two and one mini dental implant, respectively, it was shown that the small dental implant and the nearby bone experienced the least volumetric average stress. By inserting at least one small dental implant at a second molar position, the amount of stress on the abutment tooth can be reduced. An increase in the number of small dental implants relieves stress on the surrounding bone and each implant [67].
(Rungsiyakull et al.	The researcher used 3D FEA to determine the impact of different designs of the clasp on the pattern of stress distribution. They examined the differences between the conventional removable partial denture (CRPD) and the minimal implant-assisted distal extension removable partial denture (IARPD) in terms of the maximum Von Mises stress, average hydrostatic pressure, edentulous ridges, abutments, and bone tissue surrounding the implant (3D FEA). In addition, 3D FEA models of lower arches with and without bilateral minimal dental implants (MDI) at the second molar regions and RPD frameworks with Kennedy class I and Rest plate Aker clasp (RPA), Rest plate I clasp (RPI), and Akers but no clasp component were developed using 3D FEA, the stress was measured when a 200N bilateral vertical load was applied on either side of the distal extension zones.	The stress concentration of IARPD with the RPI clasp design was greater on the lingual surfaces of the abutment teeth, MDI, and bone tissue surrounding the implant compared to the other designs, which were visible on the supporting structures distally. The highest stress of Von Mises on the root surface of the abutment was reduced when MDIs were added to the RPDs. Following the CRPD and IARPD with the Akers clasp design, the designs with the RPA and RPI clasps placed second and third, respectively. The hydrostatic pressure average was similar across all groups. The results of the numerical analysis show that using the distal extension base RPD design, which includes an infra-bulge retentive clasp with mesio-occlusal rests and assistance on either side of the distal extension base from miniature dental implants attached with locator attachment, results in less stress being placed on the supporting structures. For implant-assisted RPDs, the appropriate retentive clasp design should consider the retentive force's size as well as how effectively each retentive clasp design will sustain supporting structures over time [68].

6. Conclusions

1. For many years, the FEA approach has been employed in medicine and is frequently used in the study of dental implantology. It is a useful tool for permitting endless replication of studies that difficult to be duplicated clinically in one-to-one circumstances in various settings.
2. As implant technology advances, there is no perfect implant-abutment link or dental implant design. Implant manufacturers change their general architecture and connections based on judged therapeutic benefits.
3. Similar results employing in vitro procedures have enhanced many researchers' confidence in the technique. However, it should be noted that with today's technology, dynamically transferring all the nuances of natural environments to a computer model is not possible. As a result, clinical trials are required to validate FEA results on the living biomechanics part.
4. It allows for the conduct of research without raising ethical issues, many novel treatments can be investigated and developed without putting the patient at risk.
5. The regions of applicability and limitations can be better understood by comparing new and old treatment modelling. There are various advantages to FEA research over clinical, pre-clinical, and in-vitro studies.

6. Perhaps most importantly, employing novel materials and treatment approaches that have not been properly tested will not harm humans. However, clinicians must keep in mind that all of these applications are being run on a computer, with significant limitations and assumptions that will surely affect how applicable the results are to a real-world situation.
7. The most common issue with using FEA research is that the results are overstated; because all simulated models involve simplifications, it is best to compare them in vivo within the same study.
8. The materials and planning scenarios used in various studies may differ. The FEA results must be validated by mechanical tests, traditional clinical model assessments, and pre-clinical research.
9. It is crucial to remember that, while FEA studies are useful for clinical trials, the findings they derive are not as valuable as those from clinical research. However, before beginning biomechanical clinical studies, FEA research should be consulted to refine the variables in advance.

References

[1] Srirekha A, Bashetty K. Infinite to finite: an overview of finite element analysis. Indian Journal of Dental Research. 2010;21(3):425.
[2] Rao SS. The finite element method in engineering: Butterworth-heinemann; 2017.
[3] Alaneme KK, Kareem SA, Ozah BN, Alshahrani HA, Ajibuwa OA. Application of Finite Element Analysis for Optimizing Selection and Design of Ti-based biometallic alloys for Fractures and Tissues Rehabilitation: A Review. Journal of Materials Research and Technology. 2022.
[4] Freitas CJ. Standards and methods for verification, validation, and uncertainty assessments in modeling and simulation. Journal of Verification, Validation and Uncertainty Quantification. 2020;5(2).
[5] Chang Y, Tambe AA, Maeda Y, Wada M, Gonda T. Finite element analysis of dental implants with validation: to what extent can we expect the model to predict biological phenomena? A literature review and proposal for classification of a validation process. International journal of implant dentistry. 2018;4(1):1-14.
[6] Ahmed H. Craig's restorative dental materials. British Dental Journal. 2019;226(1):9-.
[7] Zhang X, Sun Q, Ye C, Li T, Jiao F, Gao Y, et al. Finite element analysis on mechanical state on the osteoclasts under gradient fluid shear stress. Biomechanics and Modeling in Mechanobiology. 2022:1-12.
[8] De Vree J, Peters M, Plasschaert A. A comparison of photoelastic and finite element stress analysis in restored tooth structures. Journal of oral rehabilitation. 1983;10(6):505-17.
[9] Ledley RS, Huang HK. Linear model of tooth displacement by applied forces. Journal of dental research. 1968;47(3):427-32.
[10] Reddy MS, Sundram R, Abdemagyd HAE. Application of finite element model in implant dentistry: a systematic review. Journal of pharmacy & bioallied sciences. 2019;11(Suppl 2):S85.
[11] Aumnakmanee S, Yodpiji N, Jantong N, Jongprasithporn M. Finite element analysis of dental implant prosthetics. Materials Today: Proceedings. 2018;5(3):9525-34.
[12] Rakhshan V, Sforza C, Vucinic P, Vitalariu AM, De Menezes M. Advanced digital dentistry. Hindawi; 2018.
[13] Farah J, Craig R, Meroueh K. Finite element analysis of a mandibular model. Journal of oral rehabilitation. 1988;15(6):615-24.
[14] Daegling DJ, Hylander WL. 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. 2000;112(4):541-51.
[15] Erden S, Yayla P. Finite Element Stress Analysis of Airplane Seat. European Mechanical Science. 2021;5(1):6-13.
[16] Hughes TJ. The finite element method: linear static and dynamic finite element analysis: Courier Corporation; 2012.
[17] Siegele D, Soltesz U. Numerical investigations of the influence of implant shape on stress distribution in the jaw bone. International Journal of Oral & Maxillofacial Implants. 1989;4(4).
[18] Kang X, Li Y, Wang Y, Zhang Y, Yu D, Peng Y. Relationships of stresses on alveolar bone and abutment of dental implant from various bite forces by three-dimensional finite element analysis. BioMed research international. 2020;2020.
[19] Maurer P, Holweg S, Schubert J. Finite-element-analysis of different screw-diameters in the sagittal split osteotomy of the mandible. Journal of cranio-maxillofacial surgery. 1999;27(6):365-72.
[20] Jafari A, Shetty KS, Kumar M. Study of stress distribution and displacement of various craniofacial structures following application of transverse orthopedic forces—a three-dimensional FEM study. The Angle Orthodontist. 2003;73(1):12-20.
[21] Alkan I, Sertgöz A, Ekici B. Influence of occlusal forces on stress distribution in preloaded dental implant screws. The Journal of prosthetic dentistry. 2004;91(4):319-25.
[22] Fung Y-c. Biomechanics: mechanical properties of living tissues: Springer Science & Business Media; 2013.
[23] Tada S, Stegaroiu R, Kitamura E, Miyakawa O, Kusakari H. Influence of implant design and bone quality on stress/strain distribution in bone around implants: a 3-dimensional finite element analysis. International Journal of Oral & Maxillofacial Implants. 2003;18(3).
[24] Kutz M. Mechanical Engineers' Handbook, Volume 4: Energy and Power: John Wiley & Sons; 2015.
[25] Doshi DS, Pradhan TM. A comparative evaluation of different compensating curves and power arm length during retraction of maxillary anterior teeth: A finite element study. Indian Journal of Health Sciences and Biomedical Research (KLEU). 2017;10(2):183.
[26] Welty J, Rorrer GL, Foster DG. Fundamentals of momentum, heat, and mass transfer: John Wiley & Sons; 2020.
[27] Misch CE. An implant is not a tooth: a comparison of periodontal indices. Dental Implant Prosthetics-E-Book. 2014;46:18-3.
[28] Aktı A, Kaya DI. Finite Element Analysis and Application in Dental Implantology. Stress. 2020;17:18.
[29] Piotrowski B, Baptista A, Patoor E, Bravetti P, Eberhardt A, Laheurte P. Interaction of bone–dental implant with new ultra low modulus alloy using a numerical approach. Materials Science and Engineering: C. 2014;38:151-60.
[30] Rho JY, Ashman RB, Turner CH. Young's modulus of trabecular and cortical bone material: ultrasonic and microtensile measurements. Journal of biomechanics. 1993;26(2):111-9.
[31] Bicudo P, Reis J, Deus A, Reis L, Vaz M. Mechanical behaviour of dental implants. Procedia Structural Integrity. 2016;1:26-33.
[32] Gil F, Herrero-Climent M, Lázaro P, Rios J. Implant–abutment connections: influence of the design on the microgap and their fatigue and fracture behavior of dental implants. Journal of Materials Science: Materials in Medicine. 2014;25(7):1825-30.
[33] El-Anwar MI, El-Zawahry MM. A three dimensional finite element study on dental implant design. Journal of Genetic Engineering and

Biotechnology. 2011;9(1):77-82.
[34] Lee H-H. Finite element simulations with ANSYS Workbench 18: SDC publications; 2018.
[35] Holmgren EP, Seckinger RJ, Kilgren LM, Mante F. 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. 1998;24(2):80-8.
[36] Akça K, Çehreli MC, İplikçioğlu H. A comparison of three-dimensional finite element stress analysis with in vitro strain gauge measurements on dental implants. International Journal of Prosthodontics. 2002;15(2).
[37] Geng J, Xu W, Tan K, Liu G. Finite element analysis of an osseointegrated stepped screw dental implant. Journal of Oral Implantology. 2004;30(4):223-33.
[38] Lin C-L, Kuo Y-C, Lin T-S. Effects of dental implant length and bone quality on biomechanical responses in bone around implants: a 3-D non-linear finite element analysis. Biomedical Engineering: Applications, Basis and Communications. 2005;17(01):44-9.
[39] Lin CL, Chang SH, Chang WJ, Kuo YC. Factorial analysis of variables influencing mechanical characteristics of a single tooth implant placed in the maxilla using finite element analysis and the statistics-based Taguchi method. European journal of oral sciences. 2007;115(5):408-16.
[40] Quaresma SE, Cury PR, Sendyk WR, Sendyk C. A finite element analysis of two different dental implants: stress distribution in the prosthesis, abutment, implant, and supporting bone. Journal of Oral Implantology. 2008;34(1):1-6.
[41] Kong L, Gu Z, Li T, Wu J, Hu K, Liu Y, et al. Biomechanical optimization of implant diameter and length for immediate loading: a nonlinear finite element analysis. International Journal of Prosthodontics. 2009;22(6).
[42] Silva GC, Mendonca JA, Lopes LR, Landre Jr J. Stress patterns on implants in prostheses supported by four or six implants: a three-dimensional finite element analysis. International Journal of Oral & Maxillofacial Implants. 2010;25(2).
[43] Winter W, Möhrle S, Holst S, Karl M. Parameters of implant stability measurements based on resonance frequency and damping capacity: a comparative finite element analysis. International Journal of Oral & Maxillofacial Implants. 2010;25(3).
[44] Yeung TC, Hsu M-L, Dent M. Biomechanical Effect of a Zirconia Dental Implant–Crown System: A Three-Dimensional Finite Element Analysis. 2012.
[45] Barão V, Delben J, Lima J, Cabral T, Assunção WG. Comparison of different designs of implant-retained overdentures and fixed full-arch implant-supported prosthesis on stress distribution in edentulous mandible–a computed tomography-based three-dimensional finite element analysis. Journal of biomechanics. 2013;46(7):1312-20.
[46] Wen H, Guo W, Liang R, Xiang L, Long G, Wang T, et al. Finite element analysis of three zygomatic implant techniques for the severely atrophic edentulous maxilla. The Journal of prosthetic dentistry. 2014;111(3):203-15.
[47] Rezende CEE, Chase-Diaz M, Costa MD, Albarracin ML, Paschoeto G, Sousa EAC, et al. Stress distribution in single dental implant system: three-dimensional finite element analysis based on an in vitro experimental model. Journal of Craniofacial Surgery. 2015;26(7):2196-200.
[48] Moraes S, Pellizzer EP, Verri FR, Santiago Jr J, Silva J. Three-dimensional finite element analysis of stress distribution in retention screws of different crown–implant ratios. Computer Methods in Biomechanics and Biomedical Engineering. 2015;18(7):689-96.
[49] Bhering CLB, Mesquita MF, Kemmoku DT, Noritomi PY, Consani RLX, Barão VAR. 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. 2016;69:715-25.
[50] Gosavi S, Dhatrak P, Narkar K. Experimental stress analysis and fea of dentalimplants. Int J Res Eng Technol. 2016;5(03):328-31.
[51] Eom J-W, Lim Y-J, Kim M-J, Kwon H-B. Three-dimensional finite element analysis of implant-assisted removable partial dentures. The Journal of prosthetic dentistry. 2017;117(6):735-42.
[52] De Souza Batista VE, Verri FR, Almeida DAdF, Santiago Junior JF, Lemos CAA, Pellizzer EP. Finite element analysis of implant-supported prosthesis with pontic and cantilever in the posterior maxilla. Computer methods in biomechanics and biomedical engineering. 2017;20(6):663-70.
[53] 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. The Scientific World Journal. 2017;2017.
[54] Wazeh AM, El-Anwar MI, Atia RMG, Mahjari RM, Linga SA, Al-Pakistani LMA, et al. 3D FEA study on implant threading role on selection of implant and crown materials. Open Access Macedonian Journal of Medical Sciences. 2018;6(9):1702.
[55] Liu T, Mu Z, Yu T, Wang C, Huang Y. Biomechanical comparison of implant inclinations and load times with the all-on-4 treatment concept: A three-dimensional finite element analysis. Computer Methods in Biomechanics and Biomedical Engineering. 2019;22(6):585-94.
[56] Wu AY-J, Hsu J-T, Fuh L-J, Huang H-L. Biomechanical effect of implant design on four implants supporting mandibular full-arch fixed dentures: In vitro test and finite element analysis. Journal of the Formosan Medical Association. 2020;119(10):1514-23.
[57] De Melo Jr EJM, Francischone CE. Three-dimensional finite element analysis of two angled narrow-diameter implant designs for an all-on-4 prosthesis. The Journal of Prosthetic Dentistry. 2020;124(4):477-84.
[58] Valera-Jiménez JF, Burgueño-Barris G, Gómez-González S, López-López J, Valmaseda-Castellón E, Fernández-Aguado E. Finite element analysis of narrow dental implants. Dental materials. 2020;36(7):927-35.
[59] Tribst JPM, Dal Piva AMdO, Lo Giudice R, Borges ALS, Bottino MA, Epifania E, et al. The influence of custom-milled framework design for an implant-supported full-arch fixed dental prosthesis: 3D-FEA sudy. International Journal of Environmental Research and Public Health. 2020;17(11):4040.
[60] Tonin BSH, He Y, Ye N, Chew HP, Fok A. Effects of tightening torque on screw stress and formation of implant-abutment microgaps: A finite element analysis. The Journal of Prosthetic Dentistry. 2022;127(6):882-9.
[61] Mohamed AMA, Askar MG, El Homossany ME-MB. Stresses induced by one piece and two piece dental implants in All-on-4® implant supported prosthesis under simulated lateral occlusal loading: non linear finite element analysis study. BMC Oral Health. 2022;22(1):1-10.
[62] Bassi-Junior L, de Souza Silva RO, Dos Santos VHD, da Rocha Lourenço A, Trevizoli PV, Gaêta-Araujo H, et al. Mechanical analysis of prosthetic bars and dental implants in 3 and 4 implant-supported overdenture protocols using finite element analysis. Journal of Oral Biology and Craniofacial Research. 2021;11(3):438-41.

[63] Zupancic Cepic L, Frank M, Reisinger A, Pahr D, Zechner W, Schedle A. Biomechanical finite element analysis of short-implant-supported, 3-unit, fixed CAD/CAM prostheses in the posterior mandible. International Journal of Implant Dentistry. 2022;8(1):1-13.
[64] Huang Y-C, Ding S-J, Yuan C, Yan M. Biomechanical analysis of rigid and non-rigid connection with implant abutment designs for tooth-implant supported prosthesis: A finite element analysis. Journal of Dental Sciences. 2022;17(1):490-9.
[65] De Matos JDM, Lopes GdRS, Nakano LJN, Ramos NdC, Vasconcelos JELd, Bottino MA, et al. Biomechanical evaluation of 3-unit fixed partial dentures on monotype and two-piece zirconia dental implants. Computer methods in biomechanics and biomedical engineering. 2022;25(3):239-46.
[66] Tribst JPM, Campanelli de Morais D, Melo de Matos JD, Lopes GdRS, Dal Piva AMdO, Souto Borges AL, et al. Influence of Framework Material and Posterior Implant Angulation in Full-Arch All-on-4 Implant-Supported Prosthesis Stress Concentration. Dentistry Journal. 2022;10(1):12.
[67] Jia-Mahasap W, Rungsiyakull C, Bumrungsiri W, Sirisereephap N, Rungsiyakull P. Effect of Number and Location on Stress Distribution of Mini Dental Implant-Assisted Mandibular Kennedy Class I Removable Partial Denture: Three-Dimensional Finite Element Analysis. International Journal of Dentistry. 2022;2022.
[68] Rungsiyakull C, Rungsiyakull P, Suttiat K, Duangrattanaprathip N. Stress distribution pattern in mini dental implant-assisted RPD with different clasp designs: 3D finite element analysis. International Journal of Dentistry. 2022;2022.