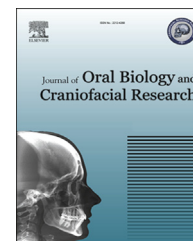


Available online at [www.sciencedirect.com](http://www.sciencedirect.com)

ScienceDirect

journal homepage: [www.elsevier.com/locate/jobcr](http://www.elsevier.com/locate/jobcr)

## Review Article

## Finite element analysis: A boon to dentistry



Shilpa Trivedi

Research Scholar, Department of Oral and Maxillofacial Surgery, FODS, King George's Medical University, Lucknow, Uttar Pradesh, India

## ARTICLE INFO

## Article history:

Received 15 November 2014

Accepted 20 November 2014

Available online 4 December 2014

## Keywords:

Finite element analysis

Implants

Trauma

## ABSTRACT

The finite element analysis (FEA) is an upcoming and significant research tool for biomechanical analyses in biological research. It is an ultimate method for modeling complex structures and analyzing their mechanical properties. In Implantology, FEA has been used to study the stress patterns in various implant components and also in the peri-implant bone. It is also useful for studying the biomechanical properties of implants as well as for predicting the success of implants in clinical condition. FEA of simulated traumatic loads can be used to understand the biomechanics of fracture. FEA has various advantages compared with studies on real models. The experiments are repeatable, there are no ethical considerations and the study designs may be modified and changed as per the requirement. There are certain limitations of FEA too. It is a computerized in vitro study in which clinical condition may not be completely replicated. So, further FEA research should be supplemented with clinical evaluation.

Copyright © 2014, Craniofacial Research Foundation. All rights reserved.

## 1. Introduction

The finite element analysis (FEA) is an upcoming and significant research tool for biomechanical analyses in biological research. It is an ultimate method for modeling complex structures and analyzing their mechanical properties. FEA has now become widely accepted as a non-invasive and excellent tool for studying the biomechanics and the influence of mechanical forces on the biological systems. It enables the visualization of superimposed structures, and the stipulation of the material properties of anatomic craniofacial structures.<sup>1</sup> It also allows to establish the location, magnitude, and direction of an applied force, as it may also assign stress points that can be theoretically measured.<sup>2</sup> Further, as it does not affect the physical properties of the analyzed materials it is easily repeatable.<sup>2,3</sup>

The finite element method (FEM) is basically a numerical method of analyzing stresses and deformations in the structures of any given geometry. The structure is discretized into the so called 'finite elements' connected through nodes. The type, arrangement and total number of elements impact the accuracy of the results.<sup>4</sup> The steps followed are generally constructing a finite element model, followed by specifying appropriate material properties, loading and boundary conditions so that the desired settings can be accurately simulated. Various engineering software packages are available to model and simulate the structure of interest may be implants or jawbone.

Previously, when FEA was used in dentistry, various simplified assumptions were made regarding modeling geometry, load, boundaries and material properties.<sup>5</sup> Such assumptions inevitably affected the analytical results. In the human body, there are individual variations with respect to

E-mail address: [shilpa.knp@gmail.com](mailto:shilpa.knp@gmail.com).  
<http://dx.doi.org/10.1016/j.jobcr.2014.11.008>

2212-4268/Copyright © 2014, Craniofacial Research Foundation. All rights reserved.

bone quality, quantity and shape which have an important impact on the prognosis of the implant or regenerative treatment. Recently with the advances of digital imaging systems (CT and MRI), it has become possible to extrapolate the individual specific data of bone geometry and property to an FEA model.<sup>6,7</sup> CT and MRI image bone and implant structure at microlevel in three dimensions. These patient specific “biological data based FEA” are peculiar to that patient as bone morphology and quality vary among individuals. Thus, very accurate anatomical models can be created which in turn provide reliable results.

For FEA modeling, a series of patient CT image data is binarized to build FEA model geometry consisting of both cortical and cancellous bone. Then apparent density, porosity or apparent ash density is appraised using different correlations to model the heterogenous distribution of mechanical properties. Most models consider isotropic behavior, since it is not possible to quantify the whole anisotropic structure of a bone, organ with current techniques.<sup>8</sup> The load is applied either to the implant or to the bone as required. Although, the muscle activity and craniofacial morphology affect the occlusal load in actual clinical situation, it is presently difficult to simulate individual muscle forces to FEA modeling. So, usually vertical or oblique load on the teeth or implant is used as an input load in FEA.<sup>9,10</sup>

---

## 2. FEA in implants

In Implantology, FEA has been used to study the stress patterns in various implant components and also in the peri-implant bone. Several FEA studies postulated that the stress pattern on peri-implant bone is affected by the implant number, diameter, length, thread profile, material properties of implant components and also by the quality and quantity of surrounding bone.<sup>11,12</sup>

In FEA studies to assess mechanical stress in the peri-implant bone, stresses of various kinds which are used include von Mises stress, the maximum, the minimum principal stress and the maximum shear stress. The von Mises stress is the most frequently and mainly used scalar-valued stress invariant to appraise yielding/failure behavior of various materials. The maximum principal stress is appropriate for the observation of tensile stress and the minimum gives an idea about the compressive stress. The use of principle stress is considered good for such studies as the bone has both ductile and brittle properties.<sup>13</sup>

It has been suggested that mechanical stress has an eminent role in maintaining home-ostasis of the bone<sup>14</sup> and in animal models, occlusal overload was predicted to be a risk factor for peri-implant bone loss.<sup>15</sup>

In an FEA study, Demenko et al<sup>16</sup> suggested to select the implant size, giving importance to its load bearing capacity. The long term results of mandibular implant supported overdentures suggest that loss of osseointegration without signs of infection was more common than peri-implantitis.<sup>17</sup> These investigations highlight the role of biomechanical condition on the implant–bone interface for long term success of the implants.

Finite element methods substantiate that the higher crown to implant ratio increase the risk of mechanical failure. A recent study suggested that oblique loading induced higher stress to the fixation screw, chiefly when the crown: implant ratio was 1.5:1.<sup>18</sup> The results are in consensus with Urdaneta et al<sup>19</sup> who demonstrated a significant correlation between screw loosening, fracture of prosthetic abutments, and crown height.

The success of FE modeling depends on the accuracy in simulating the geometry and surface structure of the implant, the material characteristics of the implant and jawbone, the loading and support conditions as well as the biomechanical implant–jawbone interface. The assumptions made during modeling and limitations of the software invariably result in inaccuracies. FEA has been used to study the biomechanical properties of implants as well as for predicting the success of implants in clinical condition. The principal difficulty of simulating living bone tissue and its response to applied mechanical forces is also solved to some extent with the use of advanced imaging techniques.

FEA gives an in depth idea about the stress patterns in the implant and more importantly in the peri-implant bone. This knowledge about the stress patterns and distribution will lead to improvement in implant designs and placement techniques.

---

## 3. FEA for maxillary obturator

Biomechanics of implant supported maxillary obturator prosthesis can also be studied in simulated maxillary resection models by FEA. De Saussa and Mattos<sup>20</sup> concluded that the maximum dislodgment of the obturator prosthesis increased with a decrease in the area of bone support, the number of implants, and the number of clips. Also the gingival mucosa, cancellous and cortical bone was subjected to compressive stress, which increased with the decrease in the area of bone support, implant number and the number of clips in the bar retention system. So, FEA was found to be important for predicting the success of implant supported prosthetic rehabilitation of patients after maxillectomy.

---

## 4. FEA for post and core restorations

FEA numerical simulation has also been used to improve the mechanical stability and long-term success of post-and-core restorations. Liu et al<sup>9</sup> suggested that for teeth with limited coronal dentin at the loading location, as maxillary premolars with large-scale tissue loss it was crucial to lower the oblique forces by reducing the lateral occlusal contact area and by preventing contact on the top of the facial cusp, thus protecting the remaining dentin from fracture.

---

## 5. FEA and periodontal ligament

Periodontal ligament (PDL) is a soft highly specialized connective tissue which is located between the tooth root and alveolar bone.<sup>21</sup> Its primary function is tooth support and it is

Download English Version:

<https://daneshyari.com/en/article/3151966>

Download Persian Version:

<https://daneshyari.com/article/3151966>

[Daneshyari.com](https://daneshyari.com)