Application of Finite Element Analysis in medicine

I A Magomedov¹,³, M S-U Khaliev¹ and A A Elmurzaev²

¹Faculty of information technology, Chechen State University, 32 Sheripov Street, Grozny, 364024, Russia
²Department of Technological Machines and Equipment. Grozny State Oil Technical University named after academician M. D. Millionshchikov, 100 H A Isaeva pr, Grozny, 364051, Russia
³E-mail: ismwork@mail.ru

Abstract. The article discusses the use of the finite element method in medicine. Finite element analysis has existed for a long time, and its application can cover vast areas of various fields. Using this technique in medicine is not new, but there are some problems that still have to be solved. In the work Finite Element Analysis or Method is explained. Briefly explained the application of the tool in the fields not related to medicine. A literature review is conducted on the use of Finite Element Analysis in medicine. The usage of the technique in dentistry is covered. Disadvantages and advantages of the use of the Finite Element Analysis tool in dentistry is presented in the work.

1. Introduction

Finite Element Analysis (FEA) or Finite Element Method (FEM) is mainly used in engineering industries such as biomedical, aviation, civil and structural. The main purpose of FEA is to work out stresses and displacements in the analyzed structure. Although, analysis can be done to investigate linear or nonlinear problems, as well as flow, electrostatics, vibrations etc.

FEA in the process simplifies the given structure, dividing it into small elements. The smaller the elements, the greater value of the correctness of the final results, but it also takes more time to analyze the problem (to overcome the loss of time, the Sensitive Analysis is used).

FEA technique or numerical solution has become handy and powerful tool with progress in the field of calculating machines (computers). Today, numerical solutions are used in the wide range of field, as mentioned earlier, and it is proven that they illustrate reliable results. Valid results or verification of the results are a significant part when considering applying FEA for the chosen problem to be analyzed.

FEA can be applicable in medicine. The FEM technique can be implemented to solve some arising problems in the fields of medicine such as design and performance analysis of the medicine devices, and further improvement and optimization etc. [1]. Thus, the approach gives opportunity to avoid wasting time and too much spending during analysis. As with any other technique, the tool has its drawbacks. The first and key to using technology to solve the problem are the results. If the results can not be validated, they weight no value. Consequently, a method of validation should exist. In addition, the correctness of the boundary conditions is also an important part in the analysis, therefore, a qualified and experienced worker is required [2].
There are three separate parts when performing studies using FEA. Parts can be divided as follows: preprocessing, solution and postprocessing. The first part includes modelling the structure, applying material properties, defining interaction between the assembled structure (if necessary), applying mesh and defining element type, constrains and application of force (pressure, twist, gravity etc.). The second part is the solution to complete analysis without errors. The postprocess collects the results of the solution and brings them together to create a presentable view that can be understood.

There are several powerful software packages for FEA. The followings are utilized as a Finite Element Analysis tool: ANSYS, Abaqus, SOLIDWORKS, Invertor Nastran etc.

2. Literature review
Application of the Finite Element Method in dentistry described in the following paragraph. The technique is mainly used in engineering fields, but it easily can be translated to analysis biomechanical properties of materials. The FEA has existed for some time and has found application in the field of medicine, where it is used to predict the stresses and strains distribution on teeth, implants and other neighbouring bones. The approach solves the stress and strain distribution problems that can not be properly solved in vivo, such as the areas between the implant and cortical bone, denture and gingiva, or around the apex of the implant in trabecular bone. The application of Finite Element Method technique in the denticity can be classified into the following bullet points.

- Study of improved shape and design of fillings, crowns, dental implants, removable dentures, dental bridges, etc.
- Examination of mutual interaction of stomatognathic system supporting structures.
- Study of residual stresses which occur as consequence of mechanical and thermal extension in crowns and dental fillings.
- Research of physiological and biochemical effects of chewing forces, teeth reactions to occlusal forces, their interaction and stress concentration.
- Research and application in orthodontics.
- Research and application in implantology.

The analysis conducted by Aleksandar G [3] using the FEM to investigate dentistry problems demonstrated the following results. A numerical approach illustrated that the technique can be useful when looking into complex systems, which can not be solved in vivo. However, the article stated that the FEM should be used only for analysis and comparison of comparable cases, but not for the final conclusion.

Similarly, studies conducted by Jianping G., and Weiqi Y. on the use of FEA or FEM in implant dentistry illustrated that the need of improvement of validation and future optimization of FEA is essential. They stated that the difficulties during simulation arises from the modelling tissues of human bones and its response to applied mechanical forces. Due to the difficulties of modeling and analyzing the problem, some simplification and assumption can be taken into account to make the solving process possible. Nevertheless, the above steps will affect the accuracy of the final results of the analysis of FEA [4].

The use of FEM goes beyond the analysis of problems in the field of dentistry and covers other problems of medicine, such as orthopedics and cardiovascular surgery. The use of the FEM technique in medicine was first introduced when the use of animals and mechanical models was not inapplicable. Decade ago, the number of elements required for a computational model was limited by the capacity of available computers. With the development and progress in computing machines the reproduction of the human body or distinct parts become possible. FEM in orthopaedics allows to understand joint behaviour and complex articular biomechanics. In a study conducted by researchers from Romania, the use of FEM for the foot and ankle joint concluded that improvement in the field of computer modeling and finite element analysis was required [5].
Recently, studies have been conducted in the field of spinal biomechanics using FEM. In the work of Driscoll M [6] the biomechanical analysis was done. The results were validated, which means that the method used was able to conduct the analysis correctly.

The application of Finite Element Analysis (FEA) for the Medical Device Design. Finite element analysis lets designers and engineers to upgrade designs while minimizing costs. Finite Element Analysis is an ideal tool in combination with the CAD software package, in terms of cost and time, for quick redesign in order to avoid a possible accident of a medical device. With advances in computer equipment and software packages, FEA can be considered as the main tool designed to manage the risks of medical equipment design processes. Its application has significantly accelerated the time required for the concept to reach the production stage of a new medical device, since improved initial prototypes reduce the time and number of cycles required for testing and development. It provides a detailed view of the various design parameters and allows one to combine and rearrange elements, materials, and shapes that may be either impossible, expensive, or time-consuming with the earlier design process, consisting of simple prototyping and testing.

3. Methodology
To illustrate the capabilities of modelling software packages and their ease of use from construction to analysis the following study was conducted. The analysis was performed using the built-in function, and the results of the work cannot be compared or verified with other researchers, since the analysis was conducted in favour of illustrating the possibilities.

For the analysis, a CAD model was built using the software package Invertor. Parts were build separately and then assembled together (figure 1). The assembly consists of: upper gum, 2 central incisors, 2 lateral incisors, 4 brackets, 4 ligatures and archwire. Materials were assigned to parts of the structure using available materials from software. Then boundary conditions were added. First, a fixture was added to the top face of the gum. Then the force was added to the archwire in the direction normal to the upper gum. Contacts have also been altered. Automatic contacts were generated (bonded contacts) and then some were changed to separation mode. Contacts between the brackets and the teeth were bonded. However, the contacts between ligature and brackets were selected to be separated. Separation contact was used in between ligature and archwire. At the end, a mesh was generated and the study was run.

4. Results
The results were gathered in software «Invertor». The aim was to analysis the structure under different conditions (different materials) in order to calculate the displacement of each tooth. Four different materials were used: rubber silicon, PBT plastic, Titanium and Silicon nitride as listed in the table 1.

![Figure 1. Assembly of the parts.](image)
Probes of displacement were taken from the central incisors and lateral incisors at the maximum (figure 2).

Analysing figure 1, one can note that the central incisors and lateral incisors are tilted outward, and the maximum displacement can be seen in the bottom, as it would be, due to fixation on the upper surface of the gum.

From table 1 it can be seen that the displacement results are not identical, as it can be for a symmetric structure with a symmetrical application of pressure. This is due to the fact that the central incisors and lateral incisors were not symmetrical in places, but instead a small slope (angle) was introduced, as in the real world. It can be seen that as the material switched to a less ductile that results of displacement for the central incisors are differ from lateral incisors. For example, central incisors displacement increases, while lateral incisors have reduction in the displacement. It also can be said that the pressure shifts from the central incisors to the lateral incisors.

![Figure 2. Deformed structure with probes of displacement.](image)

**Table 1.** Results of the analysis.

| № | Materials          | Yield strength (Psi) | Teeth (R-right, L-left) | Displacement (mm) |
|---|--------------------|----------------------|-------------------------|-------------------|
| 1 | Rubber             | 1.500*10³            | Central incisors R      | 2.881*10⁻⁷        |
|   | Silicon            |                      | Central incisors L      | 2.631*10⁻⁷        |
|   |                    |                      | Lateral incisors R      | 9.515*10⁻⁷        |
|   |                    |                      | Lateral incisors L      | 9.600*10⁻⁷        |
| 2 | PBT Plastic        | 17.992*10³           | Central incisors R      | 3.554*10⁻⁷        |
|   |                    |                      | Central incisors L      | 2.383*10⁻⁷        |
|   |                    |                      | Lateral incisors R      | 8.047*10⁻⁷        |
|   |                    |                      | Lateral incisors L      | 8.412*10⁻⁷        |
| 3 | Titanium           | 3.997*10⁴            | Central incisors R      | 2.881*10⁻⁷        |
|   |                    |                      | Central incisors L      | 2.631*10⁻⁷        |
|   |                    |                      | Lateral incisors R      | 9.515*10⁻⁷        |
|   |                    |                      | Lateral incisors L      | 9.600*10⁻⁷        |
| 4 | Silicon Nitride    | 8.847*10⁴            | Central incisors R      | 3.300*10⁻⁷        |
|   |                    |                      | Central incisors L      | 3.703*10⁻⁷        |
|   |                    |                      | Lateral incisors R      | 8.301*10⁻⁷        |
|   |                    |                      | Lateral incisors L      | 8.446*10⁻⁷        |

5. Conclusion
To conclude, the work was done to illustrated that Finite Element Analysis can be used not only in engineering fields, but also its usage in field of medicine. The work shows the application of the tool in
the field of medicine. The technique usage in the medicine can be dated back to nineties and since then the abilities of the technique exponentially increased. But still there are some problems that need to be solved to further improvement of the Finite Element Analysis technique.

References
[1] Driscoll M 2015 Biomechanical comparison of 2 different pedicle screw systems during the surgical correction of adult spinal deformities Spine Deformity 3(2) 114-21
[2] Driscoll M 2011 Biomechanical comparison of fusionless growth modulation corrective techniques in pediatric scoliosis Medical & Biological Engineering & Computing 49(12) 1437-45
[3] Aleksandar G and Dimitrije M 2017 Practical Aspects of Finite Element Method Applications in Dentistry Balkan Journal of Dental Medicine 21 70-6
[4] Geng J, Yan W and Xu W 2008 Application of the Finite Element Method in Implant Dentistry ISBN: 978-3-540- 73763-6 (Springer) 81-2
[5] Horea G and Alexandru C 2018 Medical Use of Finite Element Modeling of the Ankle and Foot Journal of interdisciplinary medicine 1-4
[6] Mark D, Jean-Marc M-T, Hubert L and Stefan P 2013 Development of a Detailed Volumetric Finite Element Model of the Spine to Simulate Surgical Correction of Spinal Deformities Hindawi Publishing Corporation BioMed Research International 1-6
[7] Rajapakse C, Kobe E, Batzdorf A, Hast M and Wehrli F 2017 Accuracy of MRI-based finite element assessment of distal tibia compared to mechanical testing Bone 108 71-8
[8] Dar F, Meakin J and Aspden R 2002 Statistical methods in finite element analysis Journal of Biomechanics 35(9) 1155-61
[9] Kamchatnov P, Umarov Kh, Chugunov A and Kazakov A 2013 Management of a patient with low back pain: the problems of efficiency and safety Neuromuscular DISEASES N.I. Pirogov Russian National Research Medical University and Chechen State University 32-8
[10] Lalonde N, Aubin C et al. 2008 Finite element modeling of vertebral body stapling applied for the correction of idiopathic scoliosis: preliminary results Studies in Health Technology and Informatics 140 111-5
[11] Driscoll M, Aubin C-E, Moreau A and Parent S 2011 Biomechanical comparison of fusionless growth modulation corrective techniques in pediatric scoliosis. Medical and Biological Engineering and Computing 49(12) 1437-45
[12] Huynh A-M, Aubin C-E, Rajwani T, Bagnall K and Villemure I 2007 Pedicle growth asymmetry as a cause of adolescent idiopathic scoliosis: a biomechanical study European Spine Journal 16(4) 523-9
[13] Manzano S, Doblaré M and Doweidar M 2015 Parameter-dependent behavior of articular cartilage: 3D mechano-electrochemical computational model Comput Methods Programs Biomed 122 491-502
[14] Kazembakhshi S and Luo Y 2014 Constructing anisotropic finite element model of bone from computed tomography (CT) Biomed Mater Eng 24 2619-26
[15] Clin J, Aubin C-E, Parent S, Sangole A and Labelle H 2010 Comparison of the biomechanical 3D efficiency of different brace designs for the treatment of scoliosis using a finite element model European Spine Journal 19(7) 1169-78