Chapter XII
Finite Element Analysis and its Application in Dental Implant Research

Antonios Zampelis
School of Applied Mathematics and Physical Sciences, Greece

George Tsamasphyros
School of Applied Mathematics and Physical Sciences, Greece

ABSTRACT

Biomechanical research has gained recognition in medical sciences. Osseointegrated dental implants, being medical devices functioning under constant load, are one of the focal points of such research. One of the most powerful tools for biomechanical research on dental implants is finite element analysis (FEA). This chapter will cope with basic elements of FEA research, the mechanical properties of bone and the various parts of dental implants, as well as delve into published literature on the subject.

INTRODUCTION

Finite element analysis (FEA) is a computer simulation technique used in engineering analysis. It uses a numerical technique called the finite element method (FEM). There are many finite element software packages available, both free and proprietary. The sophistication of this technique has rendered it an invaluable tool in biomechanical research.

The finite element analysis was first developed in 1943 by Richard Courant, who used the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, the work of M.J.Turner, R.W.Clough, H.C.Martin and L.J.Topp in 1956 established a broader definition of numerical analysis. The researchers centered on the “stiffness and deflection of complex structures”. Development of the finite element method in structural mechanics is usually based on an energy principle such as the virtual work principle or the minimum total potential energy principle.
In its applications, the object or system is represented by a geometrically similar model consisting of multiple, linked, simplified representations of discrete regions—i.e. finite elements on an unstructured grid. Equations of equilibrium, in conjunction with applicable physical considerations, such as compatibility and constitutive relations, are applied to each element and a system of simultaneous equations is constructed. The system of partial differential equations is solved for unknown values using the techniques of linear algebra or nonlinear numerical schemes, as appropriate.

In lay terms, the mathematical model is represented by a mesh geometrically identical to the object being studied. The mesh is broken down to elements. There is a set number of elements for a distinct mesh, hence the term finite element analysis. Each element is defined by points called nodes. Depending on the type of analysis, a wide variety of elements can be used, such as one-dimensional (straight or curved), two-dimensional (triangles or quadrilaterals), torus-shaped and three-dimensional (such as tetrahedrals and hexahedrals). As a force is applied, these interconnected elements start to move. Movement is defined by means of displacement of their nodes. This displacement is transformed, through the calculations, to stress and strain values (Figure 1). While being an approximate method, the accuracy of the FEA method can be improved by refining the mesh in the model using more elements and nodes.

A common use of FEA is for the determination of stresses and displacements in mechanical objects and systems. However, it is also routinely used in the analysis of many other types of problems, including those in heat transfer, fluid dynamics and electromagnetism. FEA is able to handle complex systems that defy closed-form analytical solutions.

Finite element analysis is also frequently used in biological systems, for example in orthopedics, but also for dental implants. In fact, realization of the importance of biomechanical aspects in implant dentistry have rendered FEA an essential tool in dental implant research.