CHAPTER 4

FINITE ELEMENT METHOD

4.1 Introduction

Load transmission has great influence on stress and strain values in surrounding bone and in dental materials such as dental implants, mini-bone plates and miniscrew implants (Sugiura *et al.*, 2000). Although the precise mechanisms are not fully understood (Buchter *et al.*, 2006), it is clear that there is an adaptive remodeling response of the surrounding bone to this stress (Cattaneo *et al.*, 2007). Dental equipment features causing excessive high or low stresses may contribute to pathologic bone resorption or bone atrophy (Kitamura *et al.*, 2004). Functional strain between 3,400 – 6,000 μ strain is required to maintain a normal bone remodeling rate, whereas higher or lower strain will cause a high percentage of bone resorptive surface (Melsen and Lang, 2001). Therefore, predication of the effects of loading force on dental materials and surrounding bone is important for achieving success in dental treatment. The optimal range of stress or strain values would be provided by the dental materials that are designed to reduce the effect of stress concentration.

Assessments of stress and strain in dental equipment and surrounding bone

The stress distribution in peri-implant bone has been investigated by various methods, such as photo-elastic, (Ochiai *et al.*, 2003) strain-gauge (Hekimoglu *et al.*, 2004) and finite-element analyses (Himmlova *et al.*, 2004; Tada *et al.*, 2003).

The photo-elastic modeling technique has been used to predict various biomechanical aspects of implants and implant supported prostheses (Markarian *et al.*, 2007; Ozcelik and Ersoy, 2007; Ueda *et al.*, 2004). In a photoelastic model study, it is possible to simulate the implants in bone and the shearing stresses within the loaded model (Sadowsky and Caputo, 2004). The use of this method can provide appropriate information (Brosh *et al.*, 1998). But, generally, this technique uses a two-dimensional (2-D) model that does not consider the three-dimensional (3-D) geometry of the jaws (Brosh *et al.*, 1998). In order to measure the force transfer to the bone, an optimum model simulating the complex structure of the skull is necessary

(Sevimay *et al.*, 2005). One disadvantage of the technique is that the normal stresses within the material simulating the bone cannot be detected (de Vree *et al.*, 1983). The most important insufficiency of the photo-elastic modeling method is that an isotropic photoelastic material simulates an orthropic material (bone) (Nishimura *et al.,* 1999).

Another method for investigating the stress distribution in bone uses strain gauge transducers (Akca *et al.*, 2007; Byrne *et al.*, 2006; Cehreli *et al.*, 2006). Strain gauge analysis can be used to measure relative forces on restorations connected to implants. It simulates complicated restorations, e.g. entire jaw loading when the restoration is supported by several implants. Implementation of the technique requires substantial modification of materials tested to facilitate placement of gauges (Naconecy *et al.*, 2004). But when compared with the photoelastic model method, it has a priority; it provides additional experimental information, such as stress type and strain characteristics (Cehreli *et al.*, 2004). But as with the photoelastic model, with strain gauge transducers it is impossible to simulate the complex 3-D structure of the jaws (Iplikcioglu *et al.*, 2003). Brosh (1998) has demonstrated that both of these techniques should be regarded as complementary methods.

In the past three decades, the finite element method (FEM) has become an increasingly valuable tool for the prediction of effects of stress and strain on dental equipment and on surrounding bone (DeTolla *et al.*, 2000). This method is a numerical technique for solving a complex mechanical problem by dividing the problem domain into smaller and simpler domains (elements) (Geng *et al.*, 2001). Distributions of stress and strain obtained from the solution of equilibrium equations together with applied loads and constrains (An and Draughn, 1999). However the complexity of the implant-bone system and the properties of bone are obstacles that prevent accurate approximations of exact solutions solved by the software (Olsen *et al.*, 2005).

4.2 History

The finite element method was initially introduced in the early 1960s to solve structural problems in the aerospace industry, but has since been extended to solve problems in heat transfer, fluid flow, mass transport and electromagnetic (Geng *et al.*, 2001).

In 1976, Weinstein et al were the first to use FEM in implant dentistry. They performed 2-D FEM of porous-rooted dental implants to determine the magnitude and distribution of stress in this type of dental implant. Five years later, Cook *et al.* (1981) performed (3-D) FEM. They used this method to determine the effect of implant elastic modulus on stresses in tissues around LTI carbon and aluminum oxide dental implants.

Since 1981 FEM has been increasingly applied in dental studies to calculate the stress distributions in porous rooted dental implants (Cook *et al.*, 1982), single tooth implants (Atmaram and Mohammed, 1983a, 1983b, 1983c), cylindrical implants (Meroueh *et al.*, 1987), cantilever prostheses and implant tooth retained prostheses (Akpinar *et al.*, 1996).

Stress distribution in bone correlated with implant-supported prosthesis design has been investigated by means of 2-D and 3-D FEM (Geng *et al.*, 2001). Whereas 2-D FEM is a simple and schematic model and usually used for preliminary qualitative analysis, 3-D finite element model is used for detailed qualitative analysis of the interaction between implant, tooth, ligament and bone (Romeed *et al.*, 2006). Studies comparing the accuracy of 2-D and 3-D methods demonstrated that with 3-D modeling detailed information about stress distribution in bone can be gained(Ismail *et al.*, 1987; Simon *et al.*, 1977). When 3-D FEM is compared with in vivo strain gauge measurements, the results of 3-D FEM also matched the clinical results (Benzing *et al.*, 1995).

In 2005, Motoyoshi et al. were the first to report the application of FEM in orthodontic miniscrew anchorage. They used this method to investigate effects of primary factors, especially thread pitch and presence of an abutment, on stress distribution, in an attempt to design a miniscrew implant that endures increased orthodontic force during treatment.

4.3 Advantages of FEM

If the system geometry and the mechanical properties of biologic tissues to be applied in the experiments are known, FEM should be used (An and Draughn, 1999). This numerical technique appears to overcome most of the limitations associated with the earlier experimental methods (Darbar *et al.*, 1994).

The FEM model is a mathematical model of the real object and/or phenomenon. A mathematical model can have enormous advantages over *in vivo* testing. First of all, a mathematical model is virtual and exists only in the computer. It is completely controllable. Researchers can easily change the test conditions, the model parameters and geometry, can simulate the desirable responses, and can repeat the test simulations at any time (Darbar *et al.*, 1994). Therefore, a well-tested and verified mathematical model provides researchers with a very powerful tool for analysis (Menicucci *et al.*, 2002).

A further benefit of FEM is that it is a simplified method for evaluating areas of excessive stress concentration. This method can provide clear and colorful diagrams, permitting the researcher to interpret the result from FEM easily (Huiskes and Chao, 1983; Richmond *et al.*, 2005). Moreover, this method can simulate the continuous effect produced by the parameter being evaluated (Ross, 2005).

4.4 Disadvantages of FEM

A limitation of this method is that it is not a real situation. All conditions are simulated. To overcome the lack of experimental material properties and realistic boundary conditions, detailed sensitivity or parametric tests are required, thus taking advantage of the predictive power of these numerical analyses (Korioth and Versluis, 1997).

4.5 Conclusions

FEM is an effective computational tool that has been adapted from engineering to dental biomechanics. Because of FEM, many design feature optimizations have been predicted and will be applied to potential new miniscrew systems in the future.