The Role of the Finite Element Model in Dental Implants

Daniel H. DeTolla, DDS
Sebastiano Andreana, DDS
Abani Patra, PhD
Robert Buhite, DDS
Brandon Comella, MS

KEY WORDS

Finite element methods
Dental implants
Implant design

Computer-aided design and finite element methods (FEM) have interested dental researchers because of its use in the computer simulation and design of dental implants, a process greatly facilitated by the development of new computer technology and more accurate modeling technologies. FEM allows for a better understanding of stresses along the surfaces of an implant and in surrounding bone. This will aid in the optimization of implant design and placement of the implant into the bone; it will also help when designing the final prostheses to minimize stresses. The purpose of this review is to elucidate the role of FEM and the impact of this technology in clinical dentistry in the new millennium.

All the authors are affiliated with the State University of New York at Buffalo. ***Daniel H. DeTolla, DDS****, is with the School of Dental Medicine, Department of Oral and Maxillofacial Surgery.* ***Sebastiano Andreana, DDS****, is with the School of Dental Medicine, Department of Periodontology. Address correspondence to Dr Andreana at 250 Squire Hall, School of Dental Medicine, Department of Periodontology, 3435 Main Street, Buffalo, NY 14214.* ***Abani Patra, PhD****, is with the School of Engineering, Department of Mechanical and Aerospace Engineering.* ***Robert Buhite, DDS****, is with the School of Dental Medicine, Department of Restorative Dentistry.* ***Brandon Comella, MS****, is with the Department of Mechanical and Aerospace Engineering and the School of Dental Medicine.*

Introduction

Dental implants average 1 mm of crestal bone loss around the neck of the implant within the first year of function, with subsequent loss of 0.1 mm per year.[1,2] After years of function, cumulative bone loss may become a concern, since the osseointegrated implant requires vital, nondiseased bone to prevent failure of the prosthetic implant system. Two possible etiologic reasons for crestal bone loss are bacteria leading to tissue infection (peri-implantitis) and mechanical forces that exceed ultimate stresses in the bone-implant system.

Bacterial infections are minimized by placement of sterile implants utilizing proper aseptic placement techniques and by good oral hygiene. Even when the bacterial component is controlled, there is reported a 0.1-mm decrease in alveolar bone height, which has been attributed to improper loading of the implant.[3]

Functional stress between 200 and 700 psi is reported to maintain existing alveolar bone height.[4] Stress outside this range has been reported to cause degeneration of bone tissue. Degeneration ensues if the stresses are too high, and bone atrophy occurs if the stresses are too low. Maintenance of bone levels can be achieved by proper implant and prosthesis design. This aspect can be better understood by use of computer-aided analysis and studies.

In recent years, mechanical design of dental implants have benefited from computer-aided design with associated finite element analysis. Distributions of stress and strain may be obtained from a solution of equilibrium equations (the principle that the sum of forces and moments at every point in the structure is zero, expressed as a set of par-

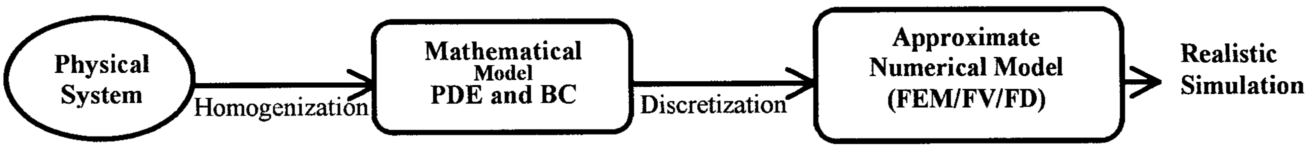


FIGURE 1. Overall computer simulation process. PDE indicates partial differential equations; BC, boundary conditions; FEM, finite element method; FV, finite volume; and FD, finite difference.

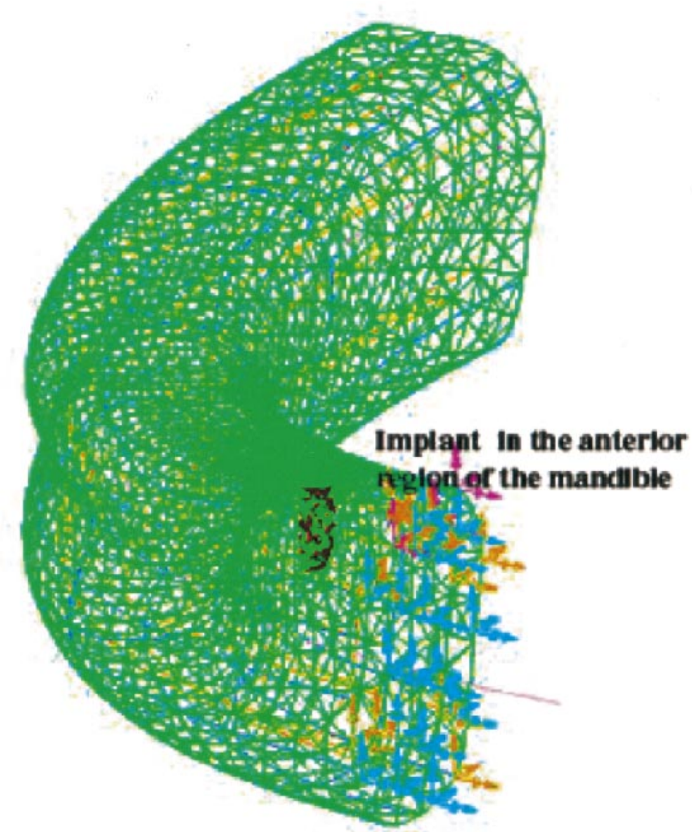


FIGURE 2. 3D finite element model. The implant is located in the anterior region of a half

tial differential equations) together with applied loads and constraints. This set of equations, together with the loads applied or constraints enforced on the structure, comprise a boundary value problem (BVP) that must be solved to obtain a detailed description of the stresses and strains. However, the complexity of implant-bone system and bone properties precludes the use of analytical techniques to solve this BVP. The alternative to intractable exact analytical solutions is to employ numerical approximation methods. One powerful numerical method is the finite element method (FEM), which can be optimized to yield accurate approximations of exact analytical solutions.

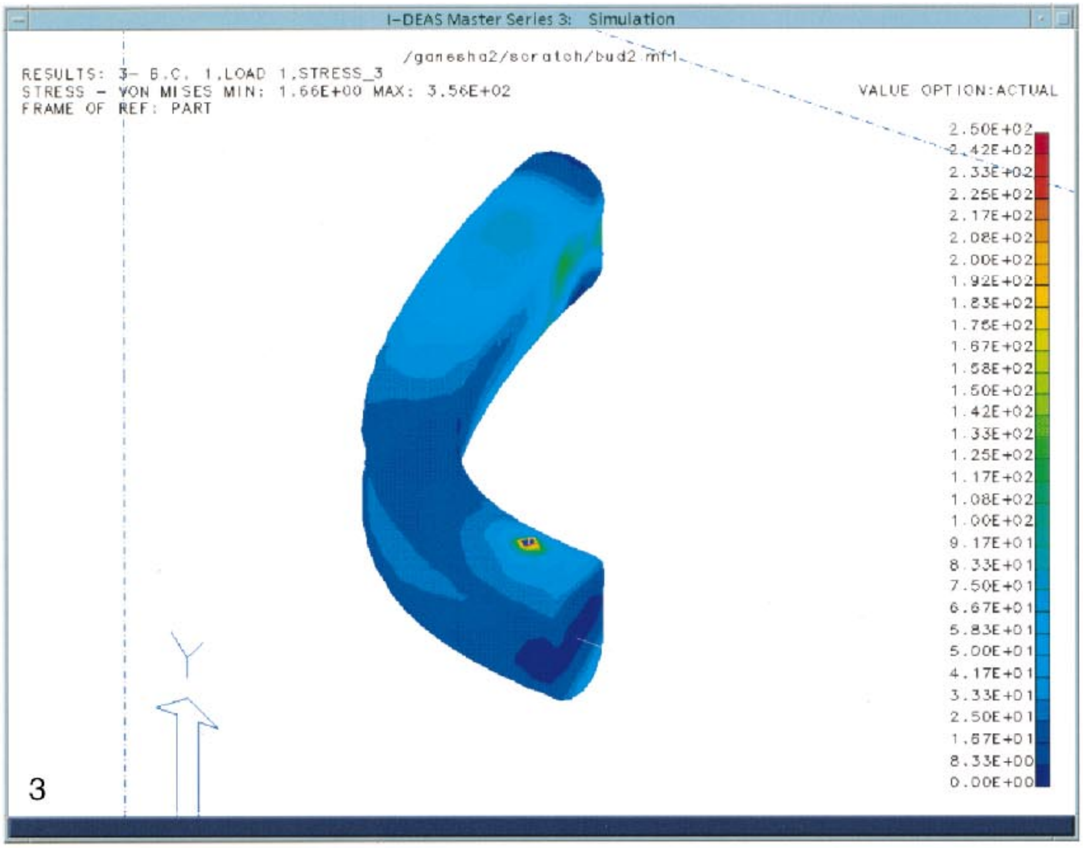
In the FEM method, the system of interest is broken into several elements, finite in number, over each of which an approximation to the solution is constructed using simple polynomial functions (*eg*, $ax + by + c$, for some a, b, c, chosen systematically to obtain a good approximation). We may illustrate the method using the problem of determining the circumference of a circle.[5] If points are placed on the circumference of the circle and line segments connect each adjacent point, then the approximate circumference would be the sum of the lengths of each line elements. We have in effect constructed linear approximations to the length over each element. It may be readily apparent that the more points and line elements there are, the more accurate the solution would be. If an infinite number of elements is used, the sum of the lengths of all elements would be equal to the analytical solution ($C = 2\pi r$). Alternatively, instead of using more line segments, we could increase the accuracy of the approximation of each segment using a quadratic approximation; we would also have much better overall accuracy.

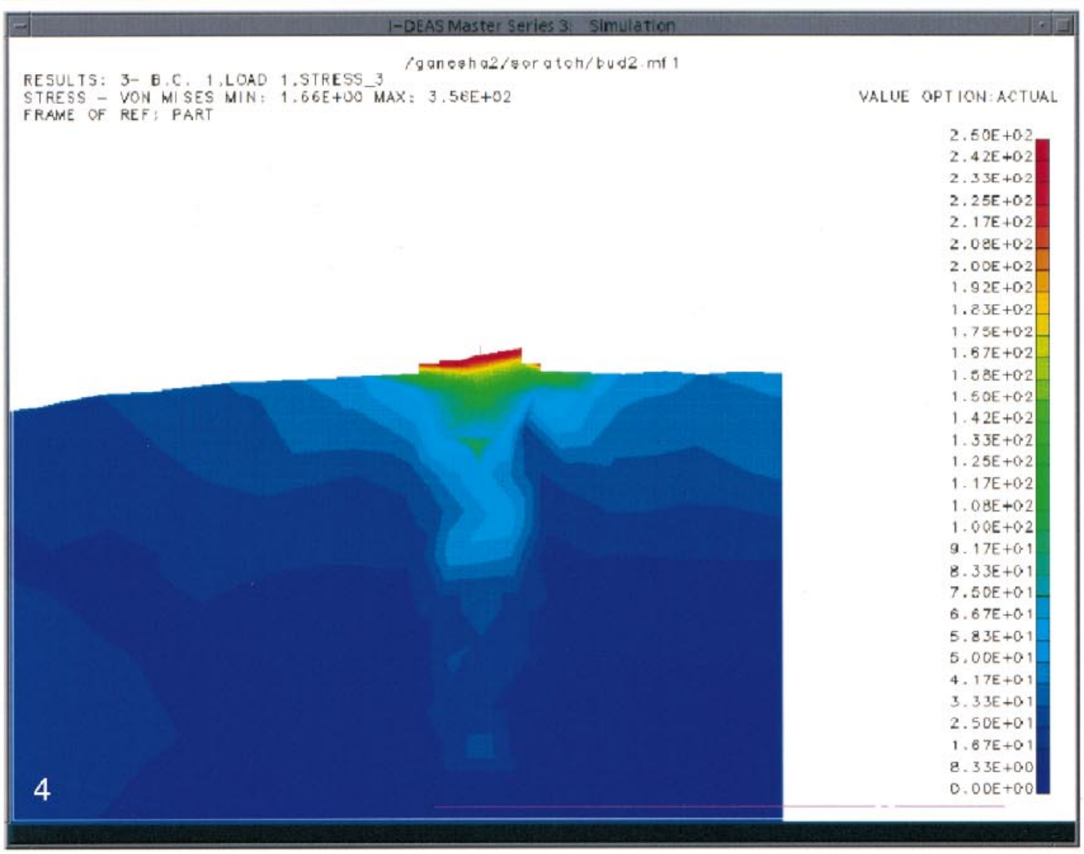
MODELING DENTAL IMPLANTS

In Fig 1, we depict the overall computer simulation process. Physical systems are modeled by partial differential equations (PDE) and boundary conditions (BC), which are resolved using discrete approximations. In each stage of the modeling process, error is introduced in the simulation. For a

→

FIGURE 3. 3D stress distribution on the implant dissipated through the elements constituting the mandibular bone.
FIGURE 4. 2D cross-sectional stress distribution along the interface of the implant and bone; notice how the cells participate individually in the process.


I-DEAS Master Series 3: Simulation
/ganesha2/scratch/bud2.mf1
RESULTS: 3- B.C. 1,LOAD 1,STRESS_3
STRESS - VON MISES MIN: 1.66E+00 MAX: 3.56E+02
FRAME OF REF: PART
VALUE OPTION:ACTUAL
2.50E+02
2.42E+02
2.33E+02
2.25E+02
2.17E+02
2.08E+02
2.00E+02
1.92E+02
1.83E+02
1.75E+02
1.67E+02
1.58E+02
1.50E+02
1.42E+02
1.33E+02
1.25E+02
1.17E+02
1.08E+02
1.00E+02
9.17E+01
8.33E+01
7.50E+01
6.67E+01
5.83E+01
5.00E+01
4.17E+01
3.33E+01
2.50E+01
1.67E+01
8.33E+00
0.00E+00
3


I-DEAS Master Series 3: Simulation
/ganesha2/scratch/bud2.mf1
RESULTS: 3- B.C. 1,LOAD 1,STRESS_3
STRESS - VON MISES MIN: 1.66E+00 MAX: 3.56E+02
FRAME OF REF: PART
VALUE OPTION:ACTUAL
2.50E+02
2.42E+02
2.33E+02
2.25E+02
2.17E+02
2.08E+02
2.00E+02
1.92E+02
1.83E+02
1.75E+02
1.67E+02
1.58E+02
1.50E+02
1.42E+02
1.33E+02
1.25E+02
1.17E+02
1.08E+02
1.00E+02
9.17E+01
8.33E+01
7.50E+01
6.67E+01
5.83E+01
5.00E+01
4.17E+01
3.33E+01
2.50E+01
1.67E+01
8.33E+00
0.00E+00
4

good simulation, these errors must be measured and controlled.

When one is interested in solutions of stress and strain, which are essentially related to the derivatives of deformation functions, much more accurate approximation and hence more computational power is necessary. Error can accumulate rapidly from oversimplification of the physical model, from improper boundary conditions, or from the numerical approximation methods. FEM is used extensively in industry to study stress and strain relationships before products are manufactured. This method enables simulation of complex dynamic physical systems by constructing approximate numerical solutions that describe the response of any system to applied loads.[6]

Early attempts at modeling dental implants resulted in unrealistic assumptions. Large errors resulted from the use of axially directed static loads; assumption of homogenous, linear, and isotropic properties of bone; assumption of perfect bonding between the bone and implant; improper boundary conditions, such as considering the inferior border of the mandible to be completely rigid and nonmovable; and modeling as two dimensional (2D). In reality, loads from mastication are dynamic and oblique relative to the occlusal surface of the implant; bone material properties are inhomogeneous, anisotropic, and nonlinear; and the interface between the implant and bone is dynamic. The approximation of loading and bone properties has been attributable to the insufficient computer capabilities for handling the calculations and insufficient bone data. These modeling errors compound, presenting a problem when interpreting the numerical solutions.[7] The sophistication in computer programming methods, computational power, and digital imaging techniques have allowed FEM to better analyze a biological structure, such as live bone, into a three-dimensional (3D) computer model more accurately.[8]

Most recent models of dental implants comprise a cortical bone region, the trabecular bone region, and a commercially pure titanium implant. This system can be analyzed using a 2D plane strain model or a complete 3D model. In the 2D system, it is assumed that out-of-plane deformations, strains, and stresses are negligible. This greatly reduces the cost of analysis, but it also introduces more error due to the artificial boundary conditions that must be assumed. The elastic modulus of the cortical and trabecular bone is usually taken as the average of several values cited in literature, where bone was assumed to be isotropic, linear elastic, homogenous, and wet/dry.[9] However, even such models lead to inconclusive data, with numerical error over 50%. Out-of-plane strains are also significant, as shown by Patra *et al.*[6] A simplified 3D finite element model of half the mandible with one implant is shown in Fig 2. From this image, we can see the discretization of the anterior portion of the mandible into a series of cells. Each cell is a finite element, and all of them together give a computerized visualization of the lower jaw. The resulting stress field is determined by FEM and shown in Fig 3. Figure 4 is the solution at a cross section through the implant-bone system derived from the full 3D simulation.

FEMs incorporating *in vivo* behavior have been used to study the different effects of axial and oblique loading on the bone-implant system.[10] The study compared nondecalcified bone-implant sections obtained from an animal after 7 weeks of function in the oral cavity with the 2D and 3D FEMs. This preliminary study shows an interesting correlation between the FEMs and histology. In another study, orthodontic forces on implants have been analyzed both histologically and with FEMs.[11]

Of particular interest is this latter study. In this report, the investigators have tried to histologically correlate *in vivo* findings with FEM. By knowing the orthodontic forces applied on osseointegrated implants in humans, Chen *et al*[11] have simulated *in vivo* conditions in the computed model. This has allowed them to identify both histologically and on the computer the areas where bone remodeling was present, areas corresponding to areas of highest stress. Interestingly, and promisingly, both the histological observations and the FEM analysis showed elevated levels of stress patterns along the implant surface in the bone in the same areas. Similar observations were made by Patra *et al*,[6] where crestal bone loss was correlated to a FEM-based stress analysis study. This preliminary study, supported by previous reports,[6,11] indicates that FEM may be used to study the pattern of stress along the interface of bone and implant and to optimize the implant design[6] in relation to stress distribution and bone quality. Clinicians will benefit from using FEM since ultimately, with this method, it will be possible to choose the most appropriate implant shape, size, design, and position according to the bone properties of that particular site.

CONCLUSION

Current research conducted by our team is moving this technology into an arena that will allow us to correlate numerical analysis with clinical practice. A new generation of custom computer programs has been written that will allow analysis of complex 3D structures that can handle hundreds of thousands of elements with complex properties. Analysis of digital image data is used to generate computer models. *In vivo* studies have recently been proposed that will allow further study of loaded implants over time, and loading patterns to empirically derive bone growth and fracture laws to be incorporated into the simulation. Multiple implant systems with prostheses have also been modeled. The availability of computer power and the increasing body of knowledge on how these systems behave will allow more accurate simulations. It may be forecasted that digital images of the maxilla or mandible, along with functional habits of the patient, may be used clinically to generate a 3D model that will suggest the implant orientation and placement, along with the desired geometry based on dynamic stress analysis.

Acknowledgments

The authors wish to thank Mr Peter Bush and the South Campus Instrumentation Center at the State University of New York at Buffalo for the technical support provided during the preparation of the manuscript.

References

1. Misch C. The implant quality scale: a clinical assessment of the health-disease continuum. In Misch C, ed. *Contemporary Implant Dentistry,* 2nd ed. St Louis, Mo: CV Mosby; 1999:21–32.
2. Adell R, Lekholm U, Rokler B, Brånemark P-I. A 15-year study of osseointegrated implants in the treatment of the edentulous jaw. *Int J Oral Surg.* 1981;10:387–416.
3. Pilliar R, Deporter D, Watson PA, Valiquette N. Dental implant design—effect on bone remodeling. *J Biomed Materials Res.* 1991;25:467–483.
4. Rieger M, Mayberry M, Brose MO. Finite Element analysis of six endosseous implants. *J Prosthet Dent.* 1990;63:667–671.
5. Reddy JN. *Introduction to the Finite Element Method.*, 2nd ed. New York: McGraw-Hill; 1993.
6. Patra A, DePaolo JM, D'Souza KS, DeTolla D, Meenaghan ME. Guidelines for analysis and redesign of dental implants. *Implant Dent.* 1998;7:355–366.
7. Ladd A, Kinney J. Numerical errors and uncertainties in finite-element modeling of trabecular bone. *J Biomechanics.* 1998;31:941–945.
8. Ulrich D, van Rietbergen B, Winans H, Ruegsegger R. Finite element analysis of trabecular bone structure: a comparison of image-based meshing techniques. *J Biomechanics.* 1998;31:1187–1192.
9. Lewinstein I, Banks-Sills L, Eliasi R. A finite element analysis of a new system(IL) for supporting an implant-retained cantilever prosthesis. *Int J Maxillofac Implants.* 1995;10:355–366.
10. Barbier L, Vander Sloten J, Krzesinski G, Schepers E, Van Der Perre G. Finite element analysis of non-axial versus axial loading of oral implants in the mandible of the dog. *J Oral Rehabil.* 1998;25:847–858.
11. Chen J, Esterle M, Roberts E. Mechanical response to functional loading around the threads of retromolar endosseous implants utilized for orthodontic anchorage: coordinated histomorphometric and finite element analysis. *Int J Oral Maxillofac Implants.* 1999;14:282–289.

--