PERIODICO di MINERALOGIA Volume 89, No. 1, 2020

Basic Principles in Finite Elements Modeling Method Applied for Metal-Ceramics Bond Analysis

I. PĂTRAŞCU*, L. T. CIOCAN*, MIHAELA BANU**
***** *Propedeutics and Dental Materials Department, Faculty of Stomatology, U.M.F. "Carol Davila", Bucharest.*
****** *Technology of Machine Building Department, Mechanics Faculty, „Dunărea de Jos" University, Galaţi.*

Abstract

With the development of biomechanics as a field of study, have been created finite elements modeling software that could be applied to a variety of problems encountered in mechanics but also in biology. In this paper we were analyzing the physic-chemical adhesion resistance level of the two components of the metal-ceramics crown by the finite elements division method. For an accurate measurement of the bond between the sintered ceramic materials and alloys, the bimaterial system bears different types of mechanical stress using specific testing devices. From these methods, we have chosen the experiment imagined by Lavine and Custer that consists of three point sustaining stress of the metal-ceramic system, due to the minimal approximation in evaluating the degree of adhesion of the metal-ceramics bond. The real mechanical experiment was translated into a virtual one, with the F.E.M. (finite element method), by projecting the original dimension of the samples in the same manner they appear in the standard accepted norms I.S.O. and D.I.N.. After probes designing and division in cubic finite elements, there were indicated the material proprieties necessary for this test (Young Elasticity Modulus, Poisson Value and Crack Resistance Level). The virtual stress was applied upon the RemaniumCS alloy – Vita Omega ceramic material system. This test simulation underlines the type of metal-ceramic adhesion failure. Also, by F.E.M., one could understand the mode of fracture propagation, the critical areas evolution and, not in the least, the degree of resistance of the metal-ceramic bond.

Purpose of Numerical Modeling in Biomechanics

Numerical modeling using finite elements is largely used from the 80's, due to the possibility of numerical simulation of the real processes and obtaining some information about the development mode of a specific process, reducing in the same time the number of experiments and the time needed to find the answers.

The numerical modeling technique was applied in engineering, solving mechanical cutting problems [4]. With the development of biomechanics as a field of study, have been created finite elements modeling software that could be applied to a variety of problems encountered in mechanics but also in biology. The principal goal of the biomechanics is that of simulating a human being capable to realize his vital functions (Modeling of Human Being Project - Japan) [1]. Biomechanics represent an interdisciplinary field of intense study, at the moment being made research regarding:

- Mathematical model creation which simulate natural phenomena from the human body;

- Imagination of the mathematical models that accurate describes the types of the materials found in human body;
- Investigation the interactions between different factors that participate to the human body functions.

One branch of biomechanics is concerned by the behavior of the new materials which substitutes dental structures (enamel, dentin, cement). Mechanics applied at this level consist in analysis of the defects, of the tearing produced by biological structure substitution with modern synthetic materials like metal-ceramics crowns and bridges, and also of the repercussions of all these upon the human body.

In the following will be presented one study test of Remanium CS Alloy- Vita Omega Ceramic Material system under the stress that evaluate the resistance level of the joint between the two components.

Numerical Modeling Using Finite Elements Method

Modeling procedures that are using finite elements method are, in present, largely spread in analyzing processes and phenomena from different fields of research and projection, and these utilizations are expected to grow up significant in the next years. The analysis procedure with finite elements is associated with deformable solids analysis, with the structures, with the heat change, with the fluids flow.

At the first time the finite elements method was developed, upon physical criteria, for the analyzing of the mechanical structure problems. It have been stated, demonstrated and recognized that this technique could be applied to a variety of problems.

In (**Figure 1**) is presented the steps in development of this procedure of numerical simulation using finite elements method.

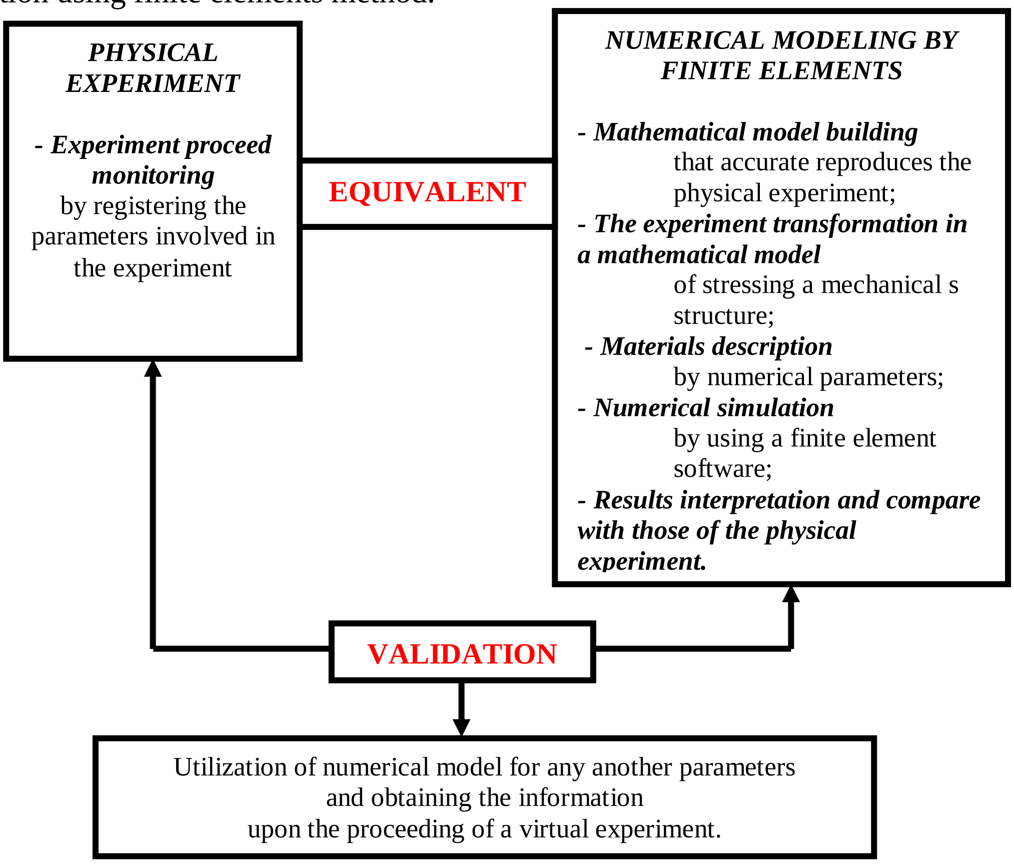

Figure 1. Basic principles in numerical modeling

The first priority problem that must be solved through finite element method is represented by tension and deformation stages finding in a mechanical structure that undergoes an outside load. This problem representation by a mathematic model implies specific hypothesis which, together, make it possible to solve the differential equations governing the mathematical model. Finite element modeling are solving this mathematical model.

Cause the answer obtained from finite element model is a numerical solution, is should reach a specific level of the results for the solution to be satisfied. If this level is not reached, than the procedure must be repeated with refined parameters of the solution, till the solution will be accepted.

The results obtained by finite elements method utilization will solve only the selected mathematical model, so all the hypothesis from this model will be reflected in the answer prediction. Using numerical modelation it is possible to obtain only the information contained by that mathematical model. From this point of view, choosing a mathematical model appropriate represent another target for numerical modeling [2].

An accurate utilization of a finite element modeling software implies the following steps:

PREPROCESSOR - building up the geometric model of the test, by creating the samples and deforming elements; - identification of the geometry with surfaces belonging to a solid corps and division of those in finite elements; - sample materials describing (plastic material) using materials coefficients: elasticity modulus, Poisson number, deformation resistance, broken resistance; - cinematic test description, by deformation process controlling, by deforming force or by speed movement of an deforming element.
PROCESSOR - proceeding the finite element modeling software, materialized by obtaining images of deformed sample;
POSTPROCESSOR - analysis of tension and deformation stages and of critical values implied in defect appearance; - forces and speed distribution analysis.

When a stress is applied upon a plastic body, it will appear two type of tensions, the normal primary tensions and cutting tensions. The effect of the two is different, in the way that principal tensions are responsible for the longitudinal deformation in same time that cutting tensions are implied in rotations of the material particles.

Time necessary for a numerical simulation is divided in time increments, by discretisation of the process time from the deformation beginning to its end, when is defined a certain criteria for finishing: or the deforming element finished the course, or a material has reached the fracture critical value, i.e. is fractured, or it has been fulfilled the maximum admissible level for that material.

Materials and Methods

Using finite elements method we were trying to analyze the metal-ceramics adhesion resistance level for a special case – the adhesion of the Vita Omega (VITA) ceramic material to Remanium CS (DENTAURUM) alloy. Vita Omega ceramic material, being a sinterizable ceramics, binds physic-chemical to metallic structure, in that case represented by modern Ni-Cr alloy specified for metal-ceramics technology – Remanium CS [10]. For quantification ceramics adhesion to metallic understructure, the double material system undergoes different types of mechanical stress using specialized testing devices.

From these methods, we have chosen the experiment imagined by Lavine and Custer that consist in three point sustaining stress of the metal-ceramic system, due to the minimal approximation in evaluating the metal-ceramics joint adhesion degree. The real mechanical experiment was translate into a virtual one, with F.E.M. (finite element method), by projecting the original dimension of the samples in the same manner they appear in the standard accepted norms I.S.O. and D.I.N..

Lavine and Custer [6] are using a metallic plate, having the ceramics sinterized on the extensible side, which is bended (**Figure 2**).

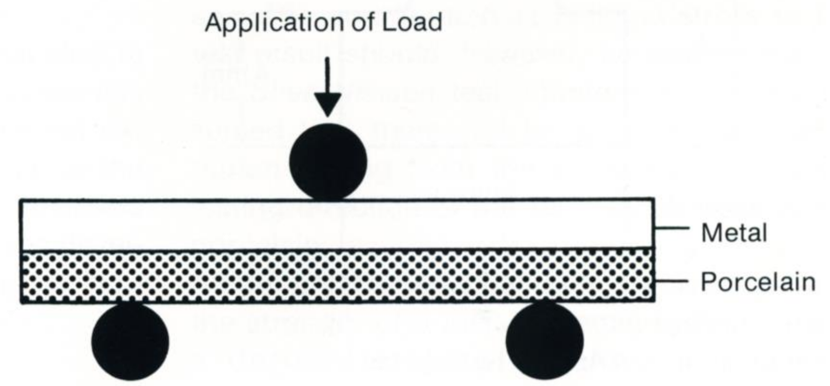

Figure 2. Transverse section: Specimen bending test diagram used by M.H. Lavine and F. Custer for determining the resistance level of metal-ceramics bond

The role of this test simulation is to realize an accurate analysis of the fracture propagation through the metal-ceramics interface level, when the biomaterial system suffers an outside stress. Due to the fact that the stress is normal orientated to the probe axis, the effect of bending this composite will the delaminating, also called the disjunction of the ceramic material from the alloy.

Starting from this physical experiment, for being possible to analyze the bimaterial behavior under tension stress [7], [8], [12], we succeed in numerical modeling this test (figure 3) using MARC 3.2. finite element modeling software [2] (**Figure 3**). Numerical modeling had principal objective that of simulation of experimental stress stages for obtaining additional information unlike those obtained by classical technique described by Lavine & Custer. These information dealt with critical area where porcelain was detached from the alloy surface, and not least the mode, the type and the direction of the fracture.

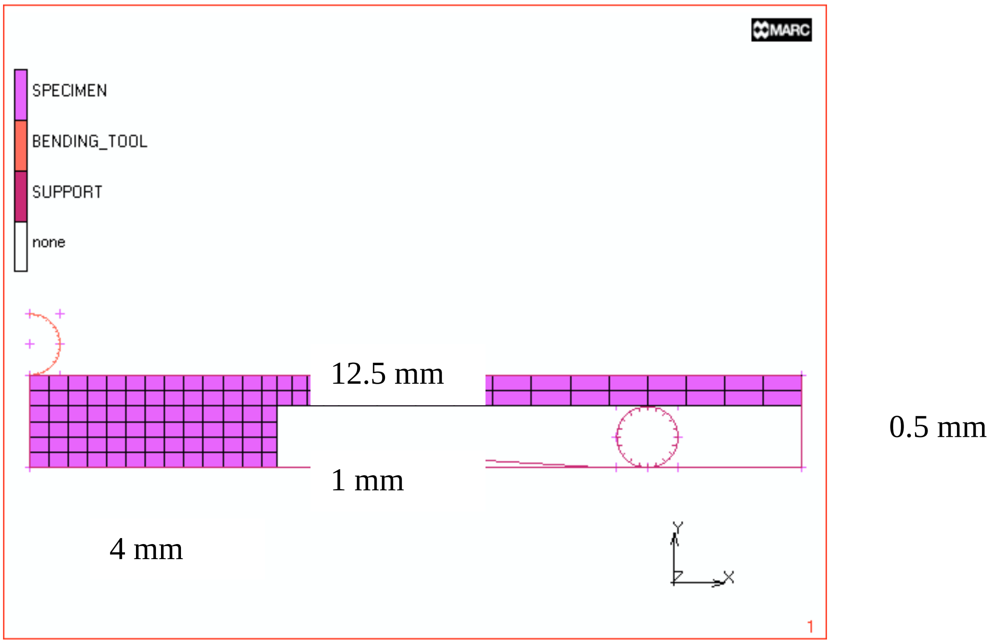

Figure 3. Probe design using finite elements method software.

In figure 3 is two-dimensional presented the scheme of deformation, on a half, because the program for numerical simulation permits the description of symmetry planes. The utilization of this method simplifies computing algorithm and substantially reduce proceeding time.

The materials couple composing the samples are REMANIUM aloy –VITA Porcelain (**Figure 4**), having materials proprieties presented in (**Table 1**).

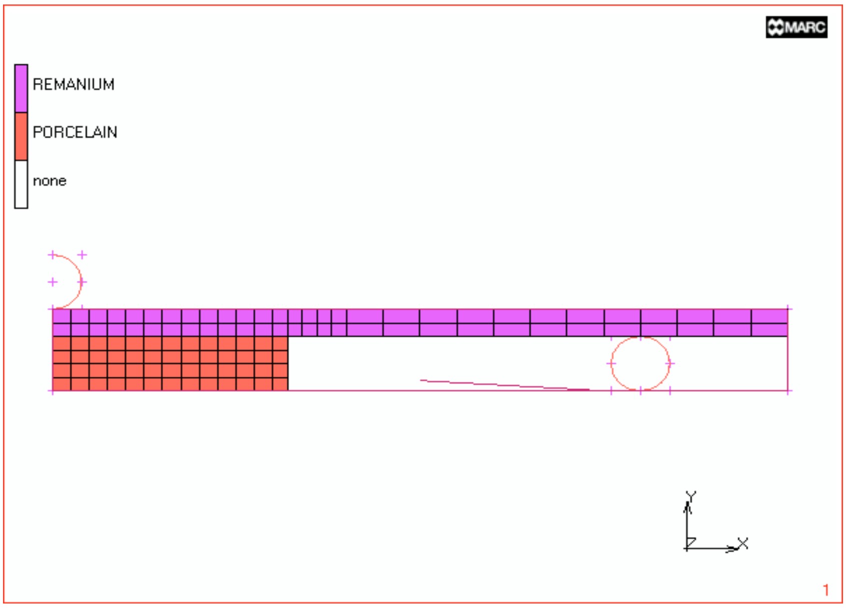

Figure 4. The F.E.M. description of the two materials that are composing the sample

Table 1.

Material	*Elasticity modulus [MPa]*	*Flowing limits [MPa]*	*Poisson number*	*Critical fracture tension [MPa]*
REMANIUM	16000	350	0.25	
PORTELAN	82800		0.19	75.9
Deforming object speed 0.1 mm/s		Time for the fracture appearance 10 s		

Results

In our evaluation, the numerical simulation had 100 increments, from those were selected for presentation 3: first one – 1st increment (**Figure 5**), the intermediate – 50th increment (**Figure 6**) and the final one – the 100th increment (**Figure 7**).

It was observed the equivalent tension map computed by Von Mises's law [13].

In figure 4 it could be noticed that applied stress induce deformation both in alloy and ceramic material. The red-green color indicates the modification of equivalent tension values in the same time that blue color is associated to null deformation stages. The deformation values could be read from the legend in the left f the image, to every color being associated, to one moment, a certain value.

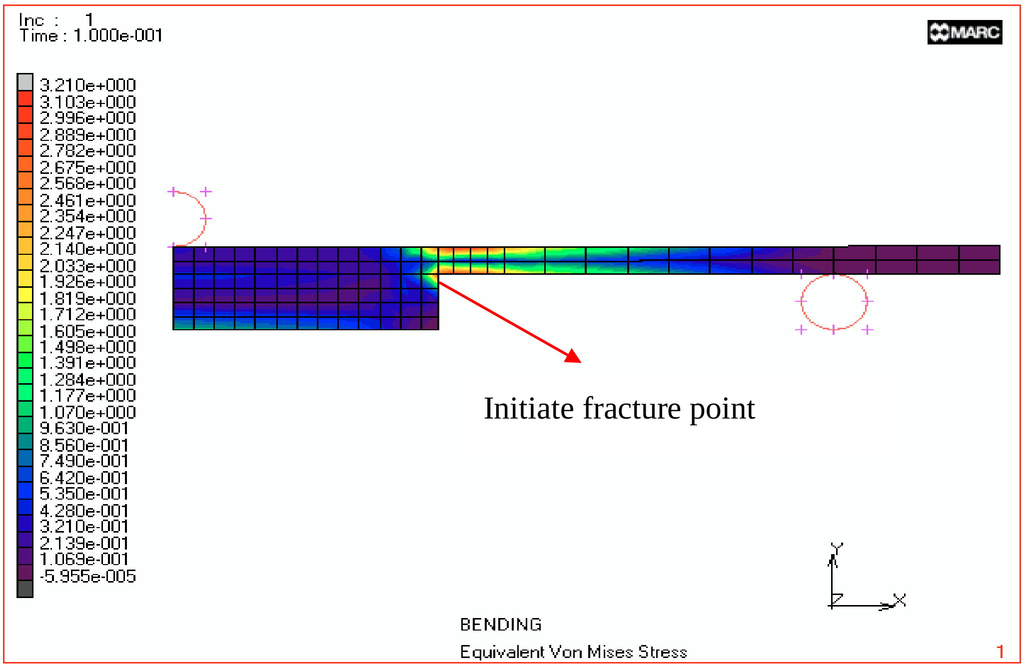

Figure 5. Tension stages at the 1st increment

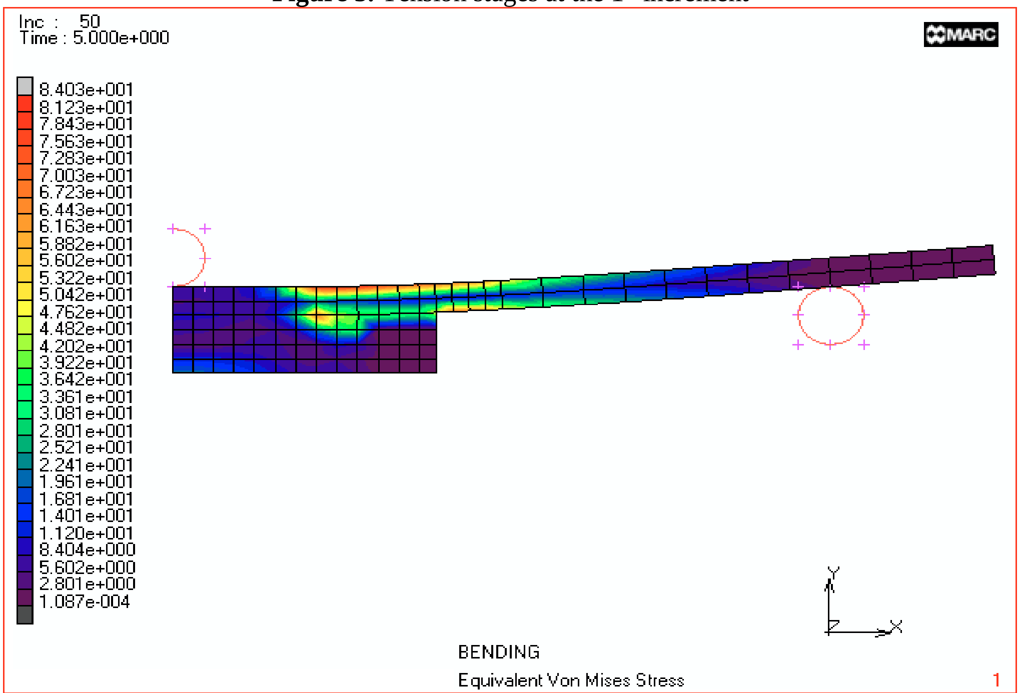

Figure 6. Tension stages at the 50th increment

In figure 6 is presented the start for the metal-ceramic fracture, when the level of deformation is beyond the acceptable limit. The acceptable limit of deformation is determined by simple tensile stressing tests, for each material, resulting curves tensions-deformations. This curves are associated to a certain type of material behavior, in that way being possible to find the value that correspond to material plasticity transformation, brake and fracture in case of plating ceramics.

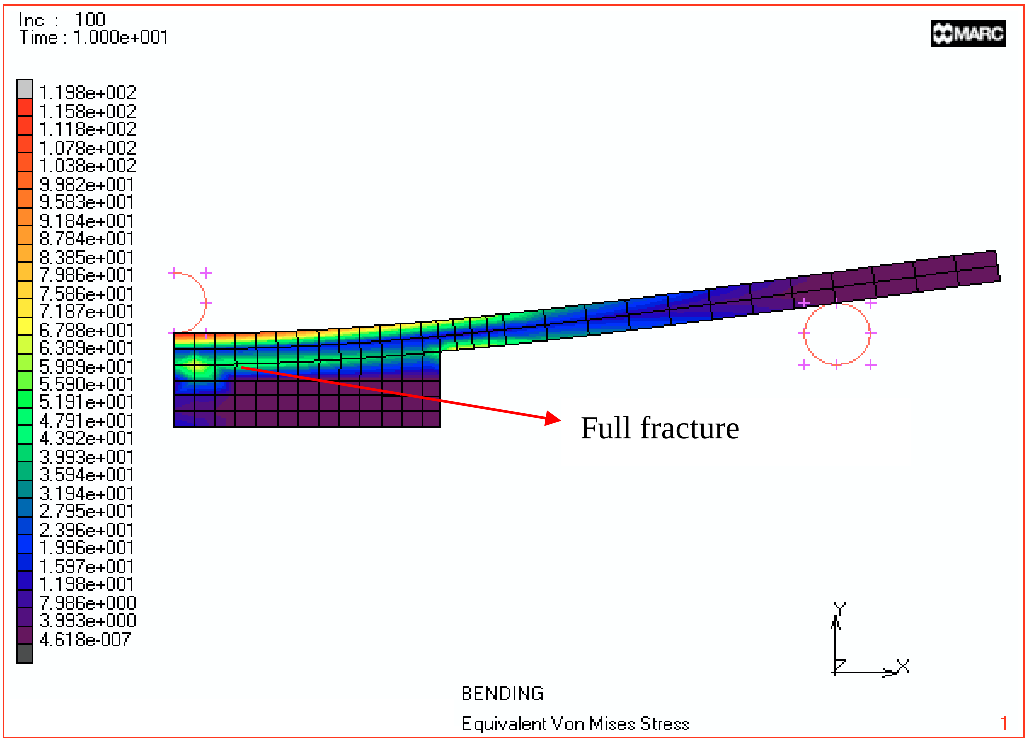

Figure 7. Tension stages at the 100th increment

Figure 7 shows the moment when the fracture is complete through the interface between ceramics and metal. It could be seen that the fracture was in direction of material minimal resistance. Deformation values indicate that sample is broken. MARC software does not permitting visualizing the effective detachment of the materials, the user must intuit this fact by analyzing the deformation or tension values legend [4]. With other words, the material is deformed to a value equal to the braking limit one.

Discussions

This test simulation bring into evidence the way of fracture propagation when the sample is stressed to a tension stage that is bent corresponding. The fracture field is not propagating suddenly through all the length of metal-ceramic joint. The fracture is first initiated and advances to a 6th part from the plated ceramic length, meet the resistance of the ceramic plate which soldieries with metal, but defeats this resistance, and advances again to one 6th of the interface length, again meet another same type of resistance, to finally succeed to reach to the symmetry plane, when practically the ceramics is completely detach.

Also, beside the qualitative aspect of fracture propagation, it is possible to analyze the values that correspond to the resistance limit level, very important fact in clinical utilization of dental crowns manufactured from these materials.

For the future, using this test, we will try to identify the mechanical proprieties of the intermediate third layer between ceramic material and dental alloy, known from specialized literature as hybrid oxides layer, which plays an important role in the two components adhesion.

Conclusions

Numerical simulation proposed by authors, unlike the classical Lavine & Custer physical experiment, bring into evidence the way of fracture propagation between the two components of metal/ceramics crown. Using this method, it is possible to have a complete image about the evolution of critical tension areas located to the interface, with the following remarks:

- by numerical simulation it is possible to test a largely types of materials, characterizing them by their coefficients, without actually produce them;
- finite element method modeling is an useful instrument for materials, time of testing and producing savings;
- by principium, the method permit studies of these materials abrasion under virtual stresses;
- virtual simulation method for a physical stress generates infinite possibilities for evaluate other type couple materials bond (alloy-resins, alloy-composites, titan-ceramics, etc.)

References

1. ***** *Biomechanical Conference Proceeding***, Editor Akitake Makinouchi, RIKEN, Tokyo, Japan, 1999.
2. ***** *MARC - User's Guide***, MARC Analyses Research Corporation, 1996.
3. ANUSAVICE K.J. – *Phillips Science of Dental Materials*, WBSaunders Co., 1996;
4. BATHE K. J. **-** *Finite Element Procedures***,** Prentice - Hall International, Inc., 1996.
5. FLINN R., TROJAN P. – *Engineering Materials and Their Applications*, 3rd edition, Boston, MA, Houghton Mifflin Co., 116-169, 1986;
6. LAVINE, M. H., AND CUSTER, F. - *J. Dent. Res.* 45-32 (1969).
7. MCLEAN J. W., SCED I. R. –*Trans. Brit. Ceram*., **Soc**., **5**:235 (1973).
8. MCLEAN, JOHN – *The Science and Art of Dental Ceramics*, *Quintessence Publ. Co.*, Chicago, vol II, 1980.
9. NOART R. – *Introduction to Dental Materials*, 2nd Edition, Mosby, 2002.
10. PĂTRAŞCU I. – *Tehnologia Aliajelor Dentare*, Libripress, Bucureşti, 2002.
11. PĂTRAŞCU & COLAB. – *Materiale Dentare – Lucrari Practice*, Horanda Press, Bucureşti, 2002.
12. SCED I. R., MCLEAN J. W., HOTZ P. – *J. Dent. Res*. **36**:1067 (1977).
13. JOHNSON W., MELLOR, P.B. - *Engineering Plasticity*, John Wiley & Sons, 1983.