6th International Conference
ðComputational Mechanics and Virtual Engineering ð
COMEC 2015
15-16 October 2015, Braşov, Romania

CONSTRUCTIVE SOLUTION, USING FINITE ELEMENT METHOD FOR GLASS CERAMIC TOOTH

Elena Anca Stanciu[1]
[1] Transilvania University of Brasov, Brasov, Romania, anca.stanciu@unitbv.ro

Abstract: *The paper presents a theoretical approach regarding the mechanical behavior of glass ceramic for tooth, this has inside FILTEK Z250 . As the most exposed part of the tooth is the sight hole, the research was focused upon the stress of this part, which was subjected to the action of the stored materials pressure. The results can be easily accessed and the input/output values of the required parameters may be identified at any point of the geometric domain. The replacement in the ideal physical system might identify the modeling errors.*
Keywords: *finite element method, glass ceramic, stress.*

1. INTRODUCTION FEM

Finite element method (FEM) becomes more and more a general method used for solving different types of complex problems concerning both stationary and non-stationary phenomena from all engineering fields but also in other activity and research areas.
As far as the stress and deformation are concerned we may observe that the internal mechanical work is linked to three components of the stress in 2D coordinates, the normal plane component of the stress does not involve the canceling of other strains or stresses.
From mathematical point of view, the problem is very similar to that of plane stress and deformation analysis, this is why the situation may be regarded as two dimensional.
By symmetry, the two components of the displacements in any 2D section of the body along the symmetry axis, completely defines the deformation state and obviously the stress state.
In order to control the complexity of the problem and “filter” the irrelevant aspects we need to accomplish a suitable mathematical model. This model should consider the fact that we are dealing with an anisotropic material, consisting of several layers and also that the loads and deformations along the contours are difficult to be obtained.
The internal stress and deformation field is locally influenced by the relative difference between the constituents’ properties, their size, shape and relative orientation as well as by the geometry of the repeating structures that form the glass ceramic.

2. MATHEMATICAL MODELLING WITH FEM

The main part of the process is, as shown in the diagram, the mathematical model. This is mostly an ordinary equation or a differential one, developed in space and time. A discrete model with finite elements is generated by help of the variation form of the mathematical model. This stage is called meshing. The FEM equations are solved using an equation solver that will provide a discrete solution.
For example, if the mathematical model is represented by a Poisson equation, the physical achievement can be represented by the heat conduction in a bar or a problem of electric charge distribution. This stage is not always necessary and may be eliminated. The FEM meshing can be done without any reference to the modeled process physical aspects.
The concept of error occurs when the discrete solution is replaced in the mathematical model. This is generally named checking. The solution error represents the extent to which the discrete solution does not check the discrete equations. More relevant are the meshing errors, representing the extent to which the discrete solution

does not check the mathematical model.

3. FEM ANALYSIS FOR GLASS CERAMIC TOOTH

The model was achieved using MSC Patran preprocessor/postprocessor and MSC Nastran processor. In the preprocessing stage, the finite elements geometric modeling requires the finite element model, which will be finally solvable by help of the programs kit meant for this purpose.
A finite element modeling requires the material behavior modeling, selection and personalization of finite elements, finite elements structure generation, introduction of boundary conditions and loads.
The analysis and solution of the finite element model, elaborated during preprocessing requires the preliminary setting of the solving parameters and the execution of the specific program modules. During this stage, the information and error messages occurred while the program is running should be carefully monitored.
The post processing of the results obtained after solving the finite elements model assumes the visualization of the deformed and animated state of the studied structure and also the visualization of various parameters using lists, diagrams and fields.
The generation of the geometric model using elementary entities was achieved by maintaining the continuity in the passing areas between one entity and the other.
The geometric modeling previous the meshing requires the generation of closed contours consisting of lines for plane areas or surfaces. In figure 1 we presented the detailed model geometry.

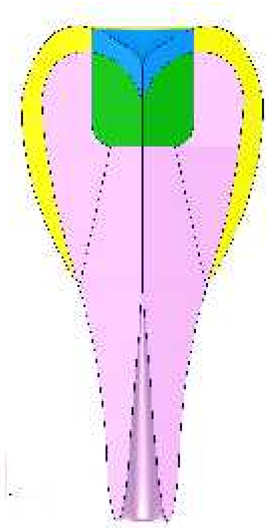

Figure 1: Geometric model of the glass ceramic tooth

Then the model will be analyzed by help of MSC Nastran processor but before running the file we need to do some previous checking in order to validate the finite elements model, as follows:
- determination of the distance between two locations or nodes;
- determination of the angle between two directions determined by three point, one of them being considered as origin;
- identification of common points;
- identification of common lines;
- identification of common nodes and joining them;
- identification of nodes belonging to a selected plane, with the possibility of moving to this plane of the nodes from the adjacent area;
- identification of the common finite elements;
- determination of a finite element distortions;
- identification of the normal in a plane finite elements group and comparing them to a given direction;
- determination of mass properties for the finite elements;
- checking the geometric boundary conditions;
- determination of the loading forces sum in a node.

Then, during post processing, the output data will be associated to both the nodes and the finite elements.

The output data corresponding to the nodes, usually include the problem unknowns, like displacements, temperatures, pressures, velocities.
The output data corresponding to the finite elements are different from one element to another, for example the internal forces, strains, deformation energy.
As material placed in the cavity (material for fillings) is Filtek Z550. These classic composite tubes are inserted into the cavities but treviscoasa is then polymerized, becoming solid.
The thickness of the ceramic crown (porcelain) is about 1 mm, we're talking about modern cearamics (Galss ceramic, zirconia) not metal cearmics. The root of remains untouched and removed from the coronal tooth.

Table 1: Materials properties used in FEA model

Material	Elastic modulus (GPa)	Poisson's ratio	Density (g/cm^3)
Glass ceramic	**70**	**0.19**	**2.40**
Zirconia	**205**	**0.19**	**2.40**

Table 2: Properties of restorative materials and tooth tissue

	Young's Modulus	Poisson's ratio	Compressive strength (Mpa)	Tensile strength (Mpa)	Shear strength (Mpa)
Composite	19 GPa	0.24	277	45	122
Amalgam	50 GPa	0.29	388	50	188
Porcelain	69 GPa	0.25	172	110	34
Enamel	80 GPa	0.30	384	10.3	90
Dentin	20 GPa	0.31	297	98.7	138
Pulp	2.07 MPa	0.45	N/A	N/A	N/A

MSC Patran model is analysed in 3D 2008R2 for tooth, containing dentin, enamel, ceramic filler above, material properties and strengths (100 N applied distributed on opposite sides of the tooth means) attached to the bottom (Figure 1).
We analyzed one case, considering the materials to dentin and enamel (enamel) of Table 2, stuffing Filtek Z250 values of material from Composite in Table 2, and pottery (above the filling) with values from Glass ceramic table 1. [2]

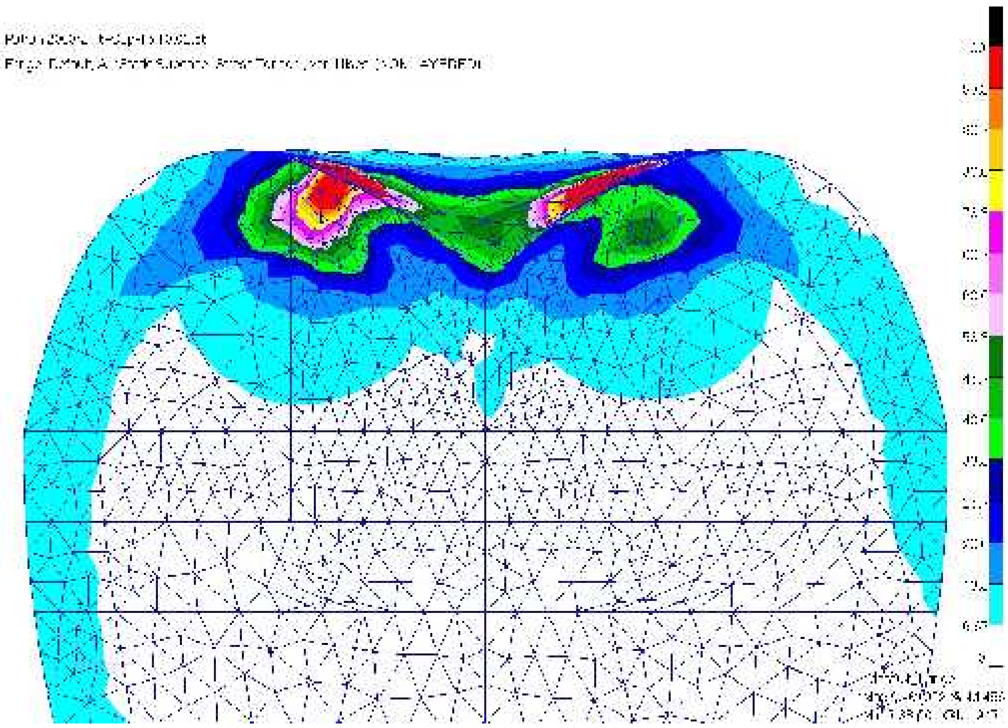

Figure 2: The maximum Von Misses stress distribution

Table 3: Results t
ooth compression stress

Properties	Results
Stiffness	1004000 N/m
Young's Modulus	390,59 MPa
Load at Maximum Load	1095,2 N
Stress at Maximum Load	38,736 MPa
Machine Extension at Maximum Load	0,761 mm
Extension at Maximum Load	0,761 mm
Strain at Maximum Load	0,06915
Percentage Strain at Maximum Load	6,915
Work to Maximum Load	461 Nmm
Load at Maximum Extension	-30,360 N
Stress at Maximum Extension	-1,0738 MPa
Machine Extension at Maximum Extension	0,766 mm
Extension at Maximum Extension	0,766 mm
Strain at Maximum Extension	0,069636
Percentage Strain at Maximum Extension	6,9636
Work to Maximum Extension	463 Nmm
Load at Break	1067,0 N
Stress at Break	37,738 MPa
Machine Extension at Break	0,761 mm
Extension at Break	0,761 mm
Strain at Break	0,069156
Percentage Strain at Break	6,9156
Work to Break	461 Nmm
Tensile Strength	38,736 MPa

3. CONCLUSION

The maximum stress was at Young's Modulus 910 MPa, for Von Misses in FEM comparation with material testing at copresion it is 390 MPa.
Following the experimental researches and also the studies based on FEM we conclude that the material thickness is oversized.

ACKNOWLEDGMENT

This paper is supported by the Sectorial Operational Programme Human Resources Development (SOP HRD), financed from the European Social Fund and by the Romanian Government under contract number POSDRU/159/1.5/S/134378

REFERENCES

[1] Guillaume Couegnata, Siu L. Fokb, Jonathan E. Cooperb, Alison J.E. Qualtroughc, , Structural optimization of dental restorations using the principle of adaptive growth, doi:10.1016/j.dental.2005.04.003
[2] Enescu,I., Stanciu, A., Finite elements, Transilvania University Publishers, 2007, ISBN 978-635-947-7.
[3] MSC/NASTRAN for WINDOWS, Version 2.0, Users manuals.
[4] Thompson, E. G., Introduction to the Finite Element Method. Theory. Programming and Applications J., New York, Wiley & Sons Publishers, 2004.