IOP Conf. Series: Materials Science and Engineering **410** (2018) 012003 doi:10.1088/1757-899X/410/1/012003

Finite element analysis of the dental crown: a case study of alumina based incisor

Pratik Samal[1], Sushree Swati Mohanty[2], Muktikanta Panigrahi[3], Saralasrita Mohanty*[4],

[1] School of Mechanical Engineering, Vellore Institute of Technology, University Vellore-632014

[2] School of Life Science, Ravenshaw University, Cuttack-753003

[3] Metallurgical Engineering Department, Gandhi Institute of Engineering and Technology (GIET), Gunupur, Rayagada, Odisha-765022

[4]School of Physical Sciences, National Institute of Science Education and Research, HBNI, Jatni-752050, India

Abstract. The purpose of this study was to evaluate the stress distribution on newly developed alumina based dental crown (incisor) for understanding its biomechanical effects by Finite Element Analysis (FEA). Alumina based dental crown was fabricated through green machining of machinable alumina cylindrical compacts by Computer Numerical Control (CNC) followed by sintering at 1550˚C. Alumina crown (Incisor) was scanned by using 3D laser scanner and modified subsequently before assembled in a 3D finite element model. The 3D model was constructed and analyzed under ANSYS environment. The model developed high stress values at the upper frontal and basal region. The large stress concentration factors and narrow profile of the upper region and the arrested base are believed to have been responsible for the high stress values. Since the stresses were lower than the compressive strength of the material, the crown is expected to withstand real-time static loading.

Keywords: Alumina, Dental crown, FEA, CNC

*Corresponding author, email-saralasrita@niser.ac.in

IOP Conf. Series: Materials Science and Engineering **410** (2018) 012003 doi:10.1088/1757-899X/410/1/012003

1. Introduction

The property of tooth mostly depends on the sign of load, loading rate, and the direction of the applied load with reference to the microstructure orientation. The effect of contact stress on the dental crown associated with the oral functions, influences the lifetime of dental restorations. Currently, study of stress distribution pattern in dental implants or crowns has become a topic of great interest due to steep rise in tooth replacements. Stress analysis of fabricated dental structures provides information related to biomechanics and skeletal anchorage system to the dentist. Further, it helps in analyzing the physical, chemical and biomedical behavior of materials being used in implant dentistry that eventually leads to predict the longevity of the dental implant. Wide range of methods have been explored for stress measurement and analysis, including strain gauge method, loading test, photo-elastic method and finite element method. However, the conventionally used methods such as strain gauge, photo-elastic methods and loading test are not feasible as they are time consuming and expensive. It is also difficult to obtain analytical solutions for the dental crown owing to the sheer complexity of dental structures. In this context, recently Finite Element Analysis (FEA) is widely used for mechanical analysis of dental implants/crown. Use of FEA though not entirely accurate, it overcomes most of the shortcomings of the conventional techniques. FEA has been widely used by contemporary researchers to study the behavior of crowns and implants in static [1] as well as dynamic [2] scenarios. FEA of crowns made of glass has been carried out by Hojjatie and Anusavice using a three dimensional model [3]. Ambulgekar and Parle have carried an FEA on a premolar tooth to study its stress pattern [1]. Jager et al. have used FEA to study the effect of varying design parameters and core materials for a ceramic dental crown [4]. Li et al. have used a finite element model to assess the reliability of ceramic dental crowns [5]. From their studies it can be concluded that FEA is a valid method to analyze complex geometries and to study dental stress distributions as it overcomes several limitations of traditional techniques. FEA can also be used to study the mechanical behavior of implants/crowns made of recently developed materials.

It is to be noted that in 2013 Mohanty et al. successfully managed to fabricate a dental crown via a novel green state CNC machining using diamond embedded tool [6]. The cylindrical alumina compacts were prepared using Protein Coagulation Casting technique. Though the material [7] used for the process has shown reasonable mechanical properties with respect to natural dental crown, however, analysis to determine the stress distribution for the crown itself has not yet carried out.

Thus, this study aims to develop the stress distribution patterns for the alumina based dental crown developed by Mohanty et al. through a Finite element model using ANSYS environment. For the analysis, two loading cases have been considered (vertical and horizontal) and the results were analyzed carefully. The magnitude of the force to be applied on the crown has been collected from a previous study [8].

2. Model Generation

Since the geometry of the incisor is not a simple one, trying to create it from scratch would have been time consuming and would have also lacked precision. Instead, an original cadaveric crown was scanned using the laser scanner. The images for the incisor were reverse engineered using reverse engineering software. The three dimensional image obtained was converted to .stl (Stereo Lithography) format which was used subsequently for the CNC machining.

Further, the prototype so manufactured was laser scanned again to obtain the model (in .stl format) used for this analysis. “Stl” file formats describe the surface geometry of the component whereas a volume/solid model was needed for the FE analysis. The stl file that was obtained was first imported to Solidworks for the generation of a solid model. The model was scaled down to the actual size of the prototype. For further work, the generated model was transferred to ANSYS Workbench as shown in figure 1 and figure 2.

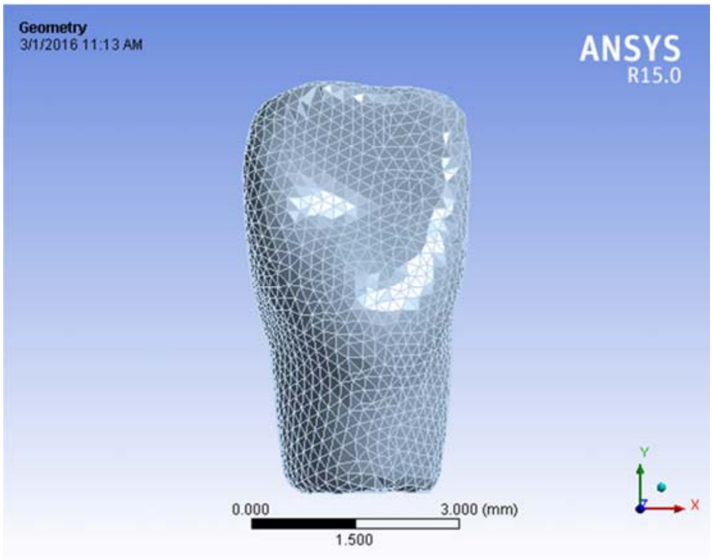


Figure 1. Front view of the dental crown in ANSYS Workbench

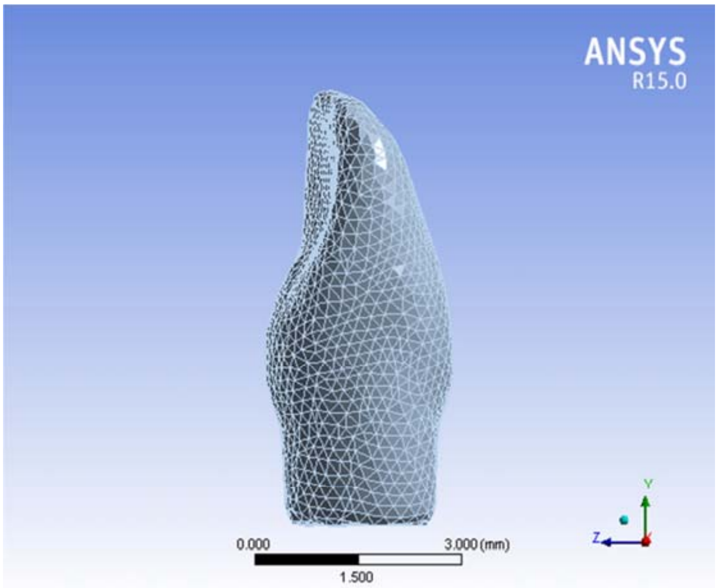


Figure 2. Side view of the dental crown in ANSYS WorkBench

3. Mechanical Properties of alumina

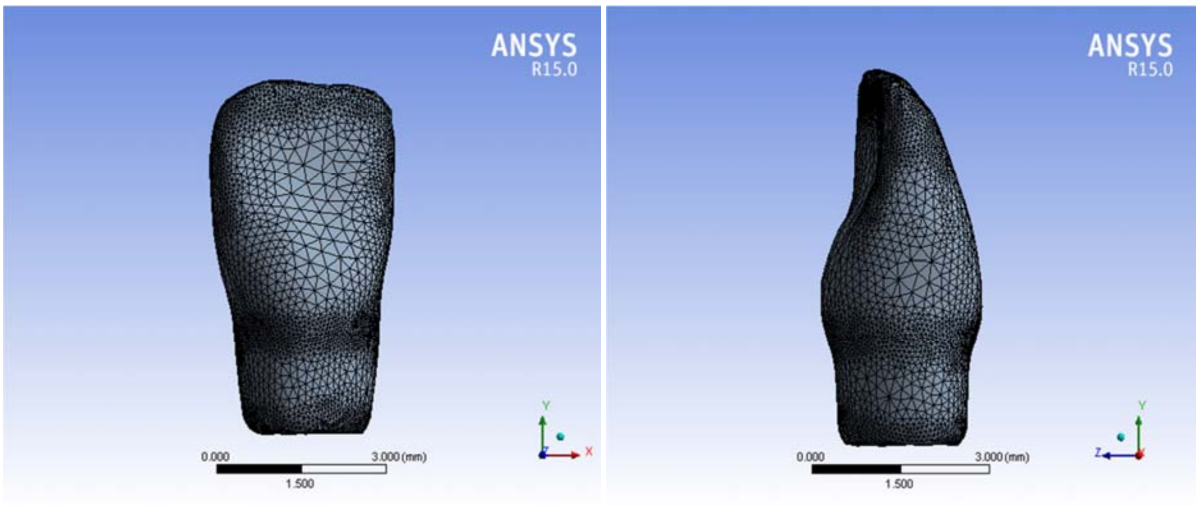
The compressive strength measurement of sintered aluimina was performed on five different cylindrical samples of diameter 13mm and 15mm in height, respectively by using **INSTRON 1344 (Capacity 500KN)**. The output obtained was, as follows.

Table 1. Mechanical properties of alumina obtained from the compressive test

Property	Value
Compressive Strength(MPa)	1037.71
Compressive stress at Yield (Offset 0.2 %) (MPa)	751.90
Young's Modulus (MPa)	23119.545
Compressive extension at Maximum Compressive load (mm)	2.69289
Compressive strain at Maximum Comp. load	17.95421 %
Poisson's Ratio	0.22

4. Meshing

In order to carry the analysis, it was first required to descretize the solid model into elements. The meshing was successfully carried out using ANSYS Workbench and the meshed model shown in figure 3(a) and figure 3(b). Element type of the mesh was SOLID 187. The node count of the meshed model was 130185, whereas the element count was 90934. The areas of the model which have irregular surfaces were selectively refined to get a better mesh quality without compromising the accuracy of the results due to increase in the mesh stiffness. A reasonable element quality of 0.83 was obtained. It must be mentioned that it was not possible to generate a perfect mesh for a geometry as complicated as that of a tooth. However, the parameters used were efficient enough to provide good results.



(a) (b)

Figure 3. (a) Front view and (b) Side view meshed model for the dental crown

5. Loads and Boundary condition

5.1. Application of boundary conditions

For applying the loads on the crown, following assumptions have been made. Human teeth have deep roots covered by gums, Hence for the analysis it was safe to assume a fixed support at the base i.e. to arrest all degrees of freedom for the base of the dental crown as indicated by the shaded blue region in figure 5 and figure 6.

5.2. Loading cases

The dental crown under study is that of an incisor. In a previous study to measure the biting forces the mean force for central incisors has been found out to be 189.3 N for normal teeth and 181 N after implantation [8]. Incisors are narrow edged teeth which are used for biting/cutting, and hence generally experience vertical loads. For the first case we have considered a vertical force 190 N in magnitude (figure 4). For the sake of completeness another case for the application of a shear force has also been considered albeit with a lesser magnitude (figure 5). The forces in their component form have been shown in table 2.

Table 2. Different loading conditions applied on the dental crown

Load Type	X component (N)	Y component (N)	Z component (N)
Vertical Force	0	-190	0
Horizontal Force	0	0	15

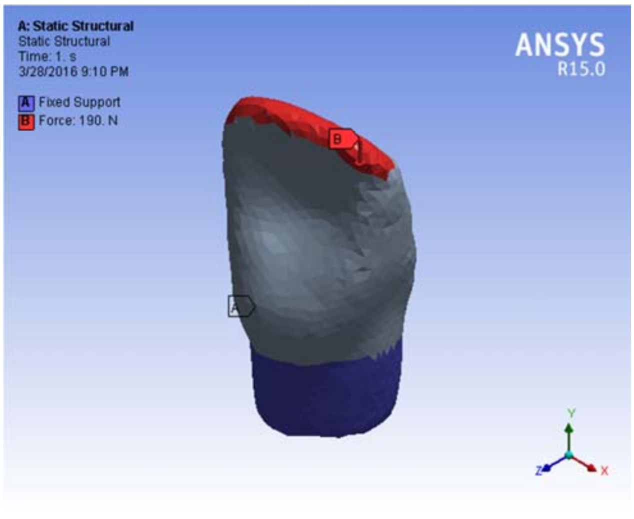


Figure 4: Application of vertical load for case-1 with fixed base

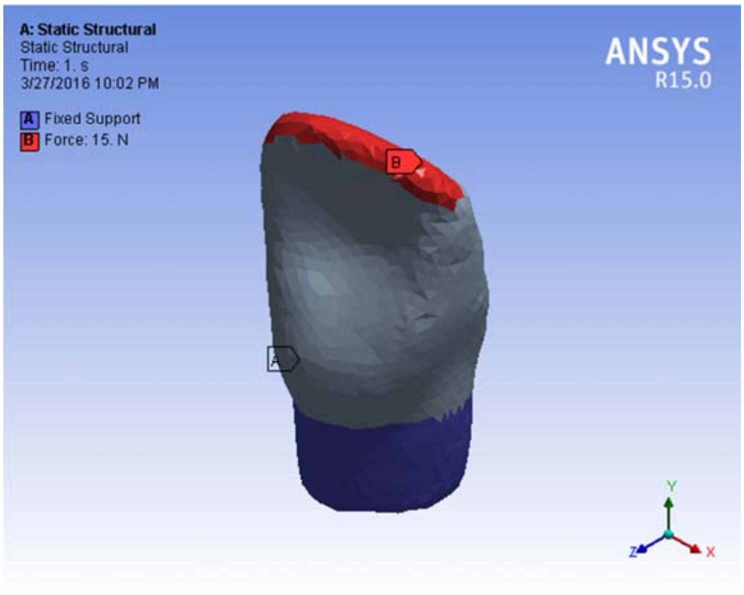


Figure 5: Application of horizontal force for case-2 with fixed base

6. Results and Discussion

The model was solved on ANSYS Workbench and the results for the Von-Mises Stress distribution were analyzed for both cases. The results have been shown in table 3.

Table 3. Maximum and minimum stress values for the two loading cases.

Load Case	Max Stress (MPa)	Min Stress (MPa)
Vertical Load	388.96	0.9198
Horizontal Load	149.92	0.0710

From figure 6 it can be clearly seen that the area just above the fixed basal region had the maximum stress values since under load, this region was unable to deform, because all degrees of freedom for the base had been arrested. But when the horizontal force was applied, the crown experienced bending about the base. As a result, in the second loading condition the frontal area was compressed, where as the rear portion was stretched which created a region of high stress near the base in the rear view as well (figure 6d). Additionally the entire front upper region of the crown under vertical loading was observed to be a region of high stress (figure 6a) for the following reasons. This region of the crown was highly irregular and had a large number of erratic contours which gave rise to large stress concentration factors. When viewed from the side we also notice that this portion was narrower compared to the base, which ultimately contributed to generation of stresses. This observation may be corroborated by a very common failure observed in dental crowns called 'chipping', wherein a portion of the tooth/crown breaks away from the main body. However the compressive strength of alumina and all brittle materials in general is very high. The stresses developed were very small compared to the compressive strength of the crown material (Table 1).

IOP Conf. Series: Materials Science and Engineering **410** (2018) 012003 doi:10.1088/1757-899X/410/1/012003

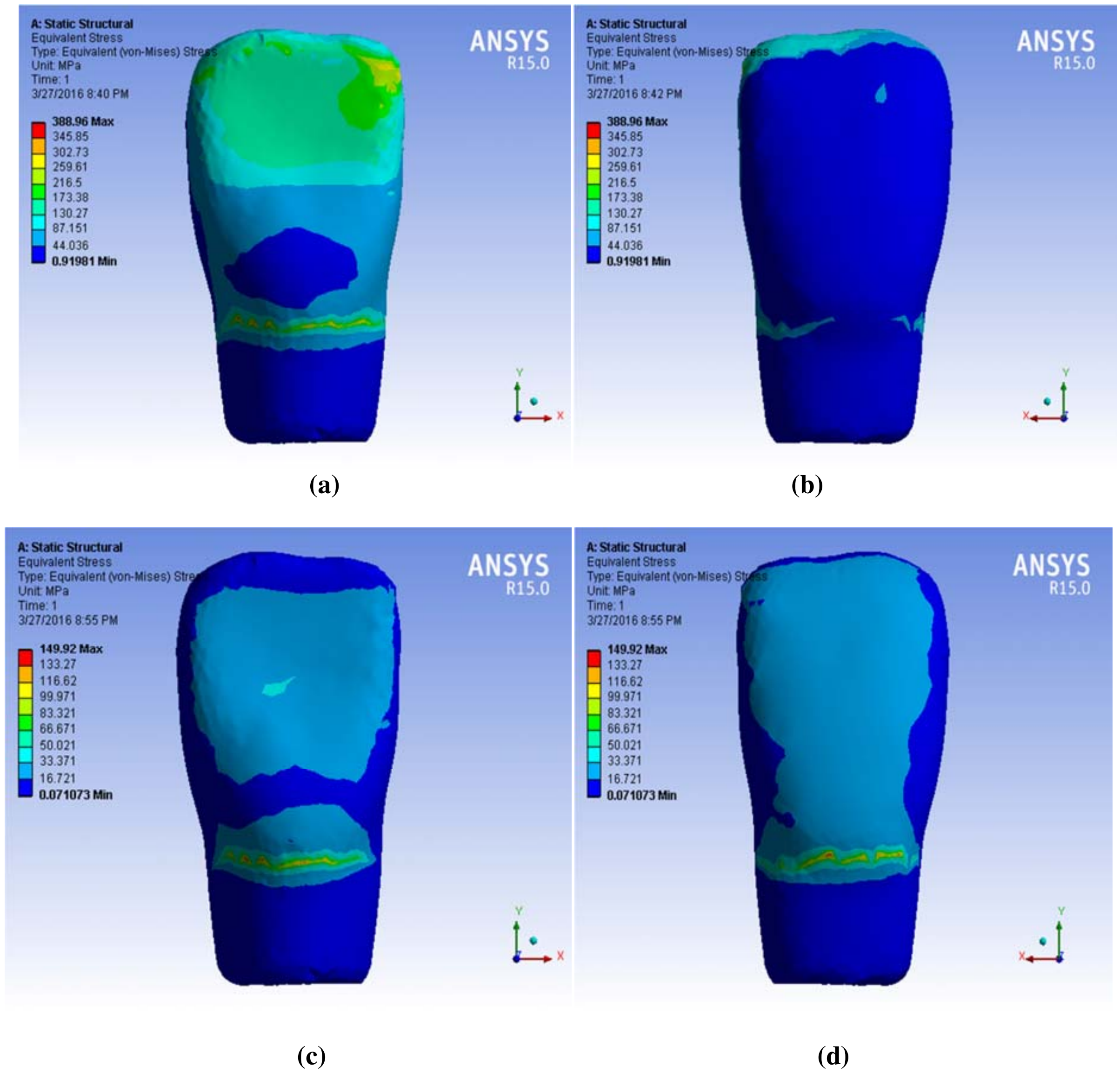


Figure 6. Von-mises stress distribution for vertical forces (a) Front, (b) Back and Horizontal forces (c) Front, (d) Back, respectively.

7. Conclusion

The finite element analysis of the alumina based incisor dental crown was successfully carried out. From this study, the maximum equivalent von-mises stress developed for the dental crown under vertical loading (190 N) was found to be 388.96 MPa. Furthermore, when a horizontal load (15N) was applied alone, the model developed a maximum stress of 149.2 MPa. For both cases, the minimum stresses developed were very small, in the range of 0.05 to 1 MPa. It was also observed that the specimen's frontal and basal region were the sites of maximum stress concentration and potential sites of crack initiation. However, the maximum stresses generated for vertical and horizontal loading were 37.48% and 14.37% of the compressive strength of alumina (1037.71 MPa). Therefore, the dental crown was safe under static loading conditions. Hence, it was concluded that the alumina based dental crown developed by Mohanty et al. could be a feasible tooth replacement in implant dentistry.

Acknowledgements

The authors would like to thank Dr. Santanu Dhara, BMTE Lab, School of Medical Science and Technology, IIT Kharagpur-721302 for assistance with the fabrication and mechanical characterization of alumina based dental crown.

References:

[1] D. Parle, A. Ambulgekar, K. Gaikwad, , Infosys HTC, 2012
[2] D. Parle, D. Desai, A. Bansal, Estimation of Individual Bite Force during Normal Occlusion using FEA, Altair Tech Conf 2013, Infosys, India
[3] B. Hojjatie, and K.J. Anusavice, 1990, Journal of Biomechanics, Vol. 23. No. 11, pp. 1157–1166
[4] N.D. Jager , P. Pallav, and A.J. Feilzer, 2005, Materials Vol. 21, pp. 242–251.
[5] Y. Li, J. Chen, J. Liu, L. Zhang, W. Wang, and S. Zhang, 2013, Biology and Medicine, Vol. 43. No. 9, pp. 1214-1220.
[6] S. Mohanty, A. Rameshbabu, S. Mandal, B. Su, and S. Dhara, 2013, Journal of Asian Ceramic Societies, Vol. 1, pp. 274–281
[7] S. Mohanty, A. Rameshbabu, and S. Dhara, 2013, Ceramics International, Vol. 39, pp. 8985–8993
[8] B.K. Biswas, S. Bag, S. Pal, Biomechanical analysis of normal and implanted tooth using biting force measurement, IJEAS, August 2013. Vol. 4, No. 2