A comparative finite element study of stress and deformation on three commonly used all ceramic fixed denture prosthesis under the influence of different bite forces

Rohit Raghavan 1*, Jency S Raj 2, Shiji M Kunjappan 3, Kiran Christopher 4, Sapna C Kambiranda 5

1 Professor & HOD, Dept. of Prosthodontics and Crown & Bridge, Royal Dental College, Kerala, India
2 Post-Graduate student, Dept. of Prosthodontics and Crown & Bridge, Royal Dental College, Kerala, India
3 Reader, Dept. of Orthodontics, Vinayaka Missions Sankaracharyar Dental College, Selam, India
4 Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Kerala, India
5 Senior Lecturer, Dept. of Prosthodontics, Coorg Institute of Dental Science, India
*Corresponding author E-mail: drrovan@gmail.com

Abstract

Aim: To simplify the complexity in design, modeling and analysis of Fixed Partial Dentures (FPD) and to evaluate the maximum stress and deformation developed using Finite-Element Method.

Materials and Methods: The specimen was created based on monoblock concept. The monoblock concept assumes that the total FPD unit, i.e., Abutment, pontic, connector and retainer together will act as a single unit. Monoblock castings do not differ from those made in segments and later welded as regard to the distribution of stresses. The Specimen of dimension 30 X 5 X 5mm were made of the three commonly used FPD materials such as feldspathic ceramics, Lithium disilicate and Zirconia. Rectangle shape was considered with notches at 1 X 450 as a close approximation of actual FPDs. The modeling was done using Solid Works 2012, and analysis was carried out using ANSYS Mechanical APDL R14.5. The maximum stress and deformations corresponding to various masticatory loads (100 N, 500 N, 600N, 800 N & 1000 N) were found out, and the results were compared with previous studies in FPD.

Results: The results revealed that the maximum principal stress and deformation are found at the centre of the pontic. Furthermore, it was found that for all masticatory loads, the maximum stress and deformation were encountered in Feldspathic ceramic and least for Zirconia.

Conclusion: A less complex model using the monoblock concept was developed to design and analyze FPD, so that the meshing complexities, computational time and cost can be reduced drastically. The results could well be considered as a benchmark in FPD-related studies.

Keywords: Fixed Partial Denture; Masticatory Loads; Finite Element Method; Monoblock

1. Introduction

There are various treatment modalities available to replace a missing posterior tooth. The conventional removable prosthesis, crown retained - Fixed Dental Protheses (FDP’s) and the recently booming implant therapy (Fradeani et al., 1997). A wide range of new materials are now available nowadays: hybrid micro filled or fiber- reinforced composites (FRC), ceramics with a high content of glass particles (i.e., lithium disilicate, glass-infiltrated zirconia, Alternatively, alumina), or high-strength ceramics (densely sintered zirconia/alumina polycrystal) to be used as frameworks for subsequent veneering or to fabricate monolithic restorations (Edelhoff et al., 2001; Wolfart et al., 2006). Due to the complex shape of FPD, experimentation and modeling of the specimen is a tedious task. So a simplified model is required to analyses and design. In this study, FPD was assumed to be a monoblock of dimension 30 x 5 x 5mm (Figure 1).

The monoblock concept assumes that the total FPD unit i.e., Abutment, pontic, connector and retainer together will act as a single unit. It helps the researcher and dental practitioner to simplify the complex FPD for the design and analysis.
occurs along plane inclined at 45° due to shearing (Rajendra, 2006). Due to this reason, The Von Mises criterion is widely used to analyses ductile materials with equal compressive and tensile strength (De et al., 1987).

This study involves analysis of brittle materials such as feldspathic ceramics, Lithium Disilicate and Zirconia. For brittle materials, authors suggest the use of the Rankine or Maximum Normal Stress criterion to evaluate the failure in dentine, using the maximum principal stress to analyse the results (Ichim et al., 2006; Nakamura et al., 2006). The theory considers the maximum normal stress as the criterion for strength.

Using this theory, modeling was done in ANSYS, and the results were analysed to study the stress distribution and deflection of the specimen. The main objectives of the study are (a) Simplify the complexity design, modeling and analysis of Fixed Partial Dentures (FPD) using monoblock concept, (b) Evaluate the maximum principal stress and deformation developed in the three commonly used Fixed Partial Dentures (FPD)’s using Finite-Element Method (FEM), (c) Arrive at benchmark for the future FPD related studies from the obtained results.

2. Numerical analysis

Three commonly used denture materials such as feldspathic ceramics, Lithium disilicate and Zirconia with similar dimensions were considered. For 3D modeling CAD software (Solidworks; Dassault Systèmes) and for the analysis part ANSYS software were used.

2.1. Modeling

Modeling of the specimen was done with the help of well-known parametric software Solid works. The first step in the parametric part modeling process is to create the two dimensional (2D) sketch of the specimen. The approach of creating two-dimensional sketches of the three-dimensional feature is the most effective and simplest way to construct solid models. After the two-dimensional, sketch was created the model is extruded to 5mm (width) to create the 3D model as shown in Figure 1. Then the model was defined in parasolid format and then fully rendered to the ANSYS software to do the analysis part.

2.2. Analysis

The mechanical properties of the specimen were obtained from different authors (Fradeani et al., 1997; Edelhoff et al., 2001; Wolfart et al., 2006; Charlton et al., 2008; Li et al., 2013; Saifang et al., 2013; Natal et al., 2014; Low, 2014) and tabulated in Table 1. The analysis was done in ANSYS Mechanical APDL R14.5. The model used in this study involves several assumptions. The model is assumed to be isotropic, homogeneous and linearly elastic.

The meshing was done (Figure 1), and each mathematical model contains 28727 nodes and 6140 elements. The size of the element is 0.0005 meter, and the type of the element is SOLID e-type. The SOLID e-type is a three-dimensional finite element with three degrees of freedom at each node (translations in the nodal x, y, and z directions).

| Table 1: Mechanical Properties of Materials Used in The Analysis |
|----------------------|------------------|------------------|
| Material             | Young’s Modulus [GPa] | Poisson’s ratio |
| Feldspathic ceramic   | 63 – 68           | 0.23             |
| Lithium Disilicate    | 95 – 100          | 0.23             |
| Zirconia              | 210               | 0.27             |

The points B and C (Figure 2) are the supporting point which is at a distance of 5mm from the adjacent edges. These points were fixed (all displacements are restricted) in all directions as the boundary conditions. The force loading during biting is reported to be 50-250N, and 500-800 N in para functions like bruxism (Kelly, 1995) moreover, it might reach to 1000 N in some conditions (Lundgren et al., 1984). In the present study 100N, 500N, 800N and 1000N (Gerami et al., 2006; Serkan et al., 2012; Mansour et al., 2014; Lakshmi et al., 2015) were selected as the extreme case of bite forces and were applied orthogonal downwards at the centre of the model (Point A).

3. Results

The FEM results are presented in terms of Principal stress values. Figure 3 shows the Principal stress distribution of Lithium Disilicate when the above-mentioned bite forces are applied, and the stress values of all mentioned ceramics are summarized in Table 2. Similarly, the total deformations of the corresponding specimen under the above specified conditions are shown in Figure 4. The total deformations of all ceramics are summarized in Table 3.

From the stress distribution (Figure 3) we can see that the stress pattern is same for all the loads (bite force), only the magnitude varies. The stress distribution is maximum at the center of the pontic, i.e. where the load is applied and minimum at both ends. In this 3D FE study, when the load is applied, it is found that the maximum principal stress is nearly same for Feldspathic ceramic and Lithium Disilicate. Slightly lesser value was obtained for Zirconia (Table 2) due to the change in the Poisson’s Ratio. From the analysis, clearly the maximum stress occurs in the center of the pontic.

From Table 3 we also understand that when the static masticatory forces are applied, the total deformation is found to be maximum for Feldspathic ceramic, and the minimum deformation for the same load is found in Zirconia. This is due to the variation in the Young’s Modulus, i.e., the larger the Young’s modulus the lesser will be the deformation.

4. Discussion

The finite-element analysis is a computer-aided mathematic technique for obtaining accurate numerical solutions used to predict the response of physical systems that are subjected to external
stress. The stress distribution in FE analysis will differ depending on the materials and properties of the material assigned to each model. FEM analysis has been extensively used to determine the stress and deformation of FPDs due to its complex structure. In the present study, the monoblock concept was introduced in FPD to simplify the complexity into the structure. Using FEM software it is possible to use a simplified model to identify the critical areas with a maximum stress and deformation.

A maximum bending stress in the FPD is induced when the load is applied at the central part of the pontic (Low, 2014). FEM analysis presented here shows that all static masticatory forces are combined together and loaded orthogonal at the centre of the pontic. It is impossible to reproduce all masticatory forces that are induced on the tooth considering a natural course. The program assumes the model as homogeneous, linear, elastic and isotropic. Due to the above assumptions, the values obtained in this study might show a little variation from the actual values. Several studies have analyzed the stress distributions in Fixed Dental Prosthesis using principal stress theory (Fischer et al., 2003; Nils et al., 2003; Eraslan et al., 2005; Paul et al., 2006; Andreia et al., 2007; Sepideh et al., 2008; Alvaro et al., 2013; Wen et al., 2011; Lakshmi et al., 2015). In one of the studies, a vertical load of 500 N was applied directed occlusally over a surface area of 5 mm2 and the results were analyzed (Lakshmi et al., 2015). In this study, the lower-right molar was fabricated with connectors of dimension 3mm x 3mm and 4mm x 4mm. The maximum principal stress observed in this case was 135.2 MPa and 147.5 MPa for Lithium Disilicate and Zirconia respectively. Additionally, the corresponding total deformation was 0.001782 mm and 0.00144 mm. The study shows that there is significant improvement in stress distribution when the dimension of connector is increased to 4mm x 4mm in both materials.

In another study (Fischer et al., 2003), three different designs as connector areas between the abutments and the pontic was considered. In the first model, (basic model) the size of the cross-sections of the connecting areas was 14 mm². In a second model, this connecting crosspiece length was again 0.1 mm as in the basic model, but the size of the cross-sections of the connectors was decreased from 14 to 8.75 mm² (2.5 x 3.5 mm). The connecting areas of a third model were of a size identical to that of the basic model (3.5 x 4.0 mm), but the length of the connecting crosspieces was increased from 0.1 to 0.5 mm. The bridges were loaded by 600 N (half-bridge model: 300 N) on the occlusal surface at 90° in the middle of the pontic component. The load was distributed onto three nodes of adjoining finite elements. The observations made in the study indicate that FE stress distribution is very similar for all 4 ceramics used as bridge materials. The maximum principal stress that occurred for all 4 ceramic bridges on the lower side of the area connecting the bridge abutment and the pontic component was 125 MPa for ZrO2.

A unit vertical load of 100N was applied at the center of the upper surface representing a 3-point-bending loading condition similar to associated standard experimental test for analyzing linear elastic stress (Nils et al., 2003). The observed maximum principal stress value in this study was 15.48 Mpa for Zirconia. Finite element analyses (FEA) were also conducted using 3D images from a micro-CT scanner for modeling Zirconia dental crowns and three-unit fixed partial dentures (Alvaro et al., 2013).

In the current study a similar FE stress distribution and deformation pattern was obtained for all the three materials considered, so the stress and deformation pattern of Zirconia and Feldspathic ceramics are not included in the paper. Comparing the Table 2 and the above discussion, the result obtained i.e., Principal stress and the total deformation in our study is maximum.

Since the study is a representation of the worst-case scenario, the values obtained in the study could be considered as the bench mark in all ceramic FPDs. The model considers the maximum load that could be applied in the specimen, and the results obtained are a representation of maximum stress and deformation values that could possibly be developed in any load case. The value of maximum stress and deformation obtained in previous studies (Ichim et al., 2006) is less than that obtained in the present study and so this monoblock concept could be considered as a reference in any FPD design optimization process.
5. Conclusions

The complex structure of the FPD makes it difficult for the researchers to design model and analyses the structure. In order to reduce these difficulties a new approach called ‘monoblock’ is introduced. This concept helps the researchers to create and analyses the FPDs easily so that the meshing complexities, computational time and cost can be reduced drastically. The results could well be considered as a benchmark in FPD-related studies. Within the limitations of the study the following conclusions can be drawn:

- A less complex model was developed to design and analyse FPD.
- Using the monoblock concept, the maximum possible values of stress and deformation was found and these values could be considered as a benchmark in future FPD related studies.
- When the load configuration is changed there is a significant change in stress and deformation values.
- The maximum stress and deformation are found at the center of the pontic region.
- The study also reinforces the fact that zirconium and Lithium Disilicate are the best suitable materials that could be used in FPD.

6. Conflict of interests

The authors declare that there are no conflict of interests regarding the publication of this paper.

References


