Numerical modeling of the fracture process in a three-unit all-ceramic fixed partial denture

Wen Kou a,∗, Shaoquan Kou b, Hongyuan Liu b, Göran Sjögren a

a Dental Materials Science, Department of Odontology, Faculty of Medicine, Umeå University, SE-901 87 Umeå, Sweden
b Department of Civil and Mining Engineering, Luleå University of Technology, SE-971 87 Luleå, Sweden

ARTICLE INFO

Article history:
Received 2 February 2006
Accepted 20 June 2006

Keywords:
Numerical modeling
Dental ceramic
Fracture
Zirconia
Fixed partial denture

ABSTRACT

Objectives. The main objectives were to examine the fracture mechanism and process of a ceramic fixed partial denture (FPD) framework under simulated mechanical loading using a recently developed numerical modeling code, the R-T2D code, and also to evaluate the suitability of R-T2D code as a tool for this purpose.

Methods. Using the recently developed R-T2D code the fracture mechanism and process of a 3U yttria–tetragonal zirconia polycrystal ceramic (Y–TZP) FPD framework was simulated under static loading. In addition, the fracture pattern obtained using the numerical simulation was compared with the fracture pattern obtained in a previous laboratory test.

Results. The result revealed that the framework fracture pattern obtained using the numerical simulation agreed with that observed in a previous laboratory test. Quasi-photoelastic stress fringe pattern and acoustic emission showed that the fracture mechanism was tensile failure and that the crack started at the lower boundary of the framework. The fracture process could be followed both in step-by-step and step-in-step.

Significance. Based on the findings in the current study, the R-T2D code seems suitable for use as a complement to other tests and clinical observations in studying stress distribution, fracture mechanism and fracture processes in ceramic FPD frameworks.

© 2006 Academy of Dental Materials. Published by Elsevier Ltd. All rights reserved.

1. Introduction

Dental ceramics are considered to have good biocompatibility, excellent translucency and appearance, and to be one of the most inert dental materials currently in use [1,2]. Despite the recent introduction of several new dental ceramic materials with reinforced core systems [3–6], the possibility of making posterior dental ceramic fixed partial dentures (FPDs) remains, however, limited. This is mainly because the ceramics are brittle and can withstand compressive stresses better than tensile stresses [7], and deformation caused by chewing and biting can induce tensile stresses leading to failure. Therefore, identification and prevention of the generation of areas with excessive stress fields is crucial for fracture resistance in all-ceramic dental restorations and for their longevity.

Since the anatomic variation of human teeth is relatively wide, as is the variation in shape and dimension of dental restorations, a number of computerized numerical methods have been used to facilitate analyses of stress distributions and to simulate mechanical loading of dental restorations [8–11]. Finite element analysis is one of the numerical simulations commonly applied in these studies. However, in a survey of the literature, no article was found that could clearly show the fracture mechanisms and processes of three-unit all-ceramic FPDs or that considered the materials’ heterogeneity. Recently, another numerical modeling code, the R-T2D code,
has been developed and has demonstrated that it could be an appropriate tool for simulating and studying the fracture processes of brittle materials [12]. The simulated results are said to explain many phenomena observed in practice better than simulations using other codes [12]. This R-T2D code was developed at the Division of Rock Engineering, Luleå University of Technology, Luleå, Sweden in order to study the processes of rock fragmentation. It is based on the rock process analysis (RFPA) code and the finite element analysis (FEA) method. Detailed descriptions of the code have been presented in papers published earlier [12,13]. Among the advantages of the R-T2D code are that it can simulate non-homogeneous materials, such as different kinds of composites and also pre-existing weaknesses. Another advantage of the R-T2D code is that Mohr–Coulomb strength criterion with tensile strength cut-off or double elliptic strength criterion are involved, making it possible to study structure failure under mechanical loading with both low and high confining pressure [12]. In addition, the stress field, the initiation and propagation of different kinds of cracks as well as the formation process of fragmentation can be shown with great clarity [12], allowing comparisons to be made with other experiments, such as determination of fracture resistance using universal testing machines [6].

Since the fracture mechanisms and processes of three-unit all-ceramic FPDs are still not clear and the newly developed R-T2D code has never been used in dentistry, the aim of the present study was to examine the fracture mechanism and process of a 3 U yttria stabilized tetragonal zirconia polycrystal (Y–TZP) FPD framework under mechanical loading using the R-T2D code and also to evaluate the suitability of the R-T2D code as a tool for this purpose.

2. Materials and methods

In the present study, the R-T2D code was utilized. Detailed descriptions of the code have been presented in papers published earlier [12,13]. The numerical simulation of the loading was simplified to a two-dimensional plane strain condition as shown in Fig. 1. The selection of the materials, the shape and dimensions of the numerical test model were made on the basis of a physical model used in a previous laboratory fracture resistance study [6]. The test model consisted of two different material parts: (i) a hot isostatic pressed (HIPed) Y–TZP ceramic for the framework, (ii) stainless steel for the abutments and the loading head (Fig. 1). The simulated model imitated the cross-section of the framework with abutments and was divided into 450 × 220 elements.

Fig. 1 – Model for numerical simulation of a 3 U dental Y–TZP FPD framework. The grey scale represents different values of Young’s modulus. The brighter the color, the higher is the value of the Young’s modulus of the element.

In the current study, the Y–TZP ceramic was assumed to be a heterogeneous material, that is, one containing impurities, voids, microcracks and other kinds of weaknesses. To characterize this heterogeneity, the R-T2D code utilized a probabilistic method in arranging the distribution of the defects and gave elements different mechanical properties, such as strength, Young’s modulus, and Poisson’s ratio. Based on earlier works, Weibull’s distribution worked satisfactorily in describing the distribution of defects in rocks and other brittle materials [14].

Since ceramics are also brittle materials, Weibull’s distribution was applied in the current simulation.

The formula for Weibull’s distribution law is

\[ \psi(\sigma) = \frac{m}{\alpha_0} \left( \frac{\sigma}{\alpha_0} \right)^{m-1} \exp \left[ -\left( \frac{\sigma}{\alpha_0} \right)^m \right] \]  

where \( \sigma \) is the element parameter that can be strength, Young’s modulus or Poisson’s ratio, \( m \) the shape parameter of the distribution curve and named homogeneous index or Weibull’s modulus. A larger \( m \) indicates a more homogeneous material. The stainless steel was assumed to be a homogeneous and ductile material with a large homogeneous index \( m \).

The other input parameters used for the numerical simulation in the present model were Young’s modulus and Poisson’s ratio. The values of the parameters used for the calculations are presented in Table 1. For Y–TZP, the values were obtained from the manufacturer’s information or from existing literature [15,16]. For the stainless steel, the values were taken from existing literature [17]. The black colored areas surrounding the framework in Fig. 1 were defined as empty.

In order to mimic laboratory situations, the loading head was applied in the centre of the top of the framework perpendicular to the framework’s ‘occlusal plane’, but five elements shifted to the right towards one of the abutments. Five ele-

<p>| Table 1 – Parameters of the Y–TZP ceramic and the stainless steel used in the present numerical simulation |
|-------------------------------------------------|------------------------------------------------|-----------------|----------------|----------------|</p>
<table>
<thead>
<tr>
<th>Materials</th>
<th>Uniaxial compressive strength (MPa)</th>
<th>Tensile strength (MPa)</th>
<th>Young’s modulus (GPa)</th>
<th>Weibull’s modulus</th>
<th>Poisson’s ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stainless steel</td>
<td>3030 [17]</td>
<td>3030</td>
<td>204 [17]</td>
<td>100</td>
<td>0.18 [17]</td>
</tr>
</tbody>
</table>

\(^a\) Manufacturer’s information.
ments corresponded to around 0.2 mm in the physical model. The reason for this experimental set-up was that when specimens are seated manually in laboratory tests, it is difficult to achieve totally balanced loading conditions. A displacement increment of 0.002 mm per step, which indicated increased loading, was then statically applied on the loading head. As soon as the framework was subjected to load, the R-T2D code started its calculation of the corresponding deformations and other results. Apart from the R-T2D code showing the fracture process step-by-step, once the stress-elements reach the failure criterion, it could also show the fracture process step-in-step. Step in step was applied when there were chain reactions in element failure within one loading step.

In the present study, compressive stress was defined with a positive value, whereas tensile stress was defined as negative. Therefore, the maximum principal stress described the compressive stress and minimum principal stress described the tensile stress. The framework deformed under loading and linear elastic relationship, i.e. Hooke’s law, was applied to describe the relationship between stress and strain until failure occurred. In the R-T2D code, Mohr–Coloumb and double elliptic strength criteria are available for judging whether the material is in an elastic state or failure. However, in the present case the confinement was not sufficiently high and, therefore, only Mohr–Coloumb strength criterion with tensile strength cut-off was applied as the failure threshold. After failure, linear damage fracture mechanics was used. More detailed descriptions can be found in earlier papers [12].

3. Results

3.1. Stress distribution and quasi-photoelastic stress fringe pattern

Photoelasticity is an experimental stress analysis method used to study the mechanics problems of elasticity using optic principles [18]. It is known that an interference fringe pattern can be seen in a model structure made of transparent plastic material in a field of polarized light if the plastic material has the characteristics of double refraction under loading [18]. The stress state of the model structure can, thus, be identified by analyzing the interference fringe pattern. A similar interference fringe pattern can be obtained numerically using the R-T2D code, the so-called quasi-photoelastic stress fringe pattern. Quasi-photoelastic stress fringe pattern is not optic interference fringes but stress divisions. Different fringes represent different stress levels.

From the quasi-photoelastic stress fringe pattern one can see the stress distribution under loading not only for transparent materials but also for opaque materials such as ceramics and steel, which makes it possible to identify the fracture-risk area and even the crack initiation location. In the present study, the fringe pattern for either maximum principal stresses or minimum principal stresses through various elements is calculated and depicted with great clarity (Fig. 2). From Fig. 2A and B the difference between homogeneous and heterogeneous materials could be identified. For example in the areas of the abutments, the photoelastic stress fringe pattern was rather clean (similar to straight lines). This is typical behavior of a homogeneous material. However, in the pontic area of the framework, the material was more heterogeneous and the fringe pattern was less distinct, due to heterogeneity-induced stress fluctuation (Fig. 2A and B). According to the simulated results, the loading stress field was not uniform in the pontic area, but rather uniform in the abutments. This can be seen in the figures for the maximum principal stress (Fig. 2A) and for the minimum principal stress (Fig. 2B). In Fig. 2A and B one can see that the compressive stresses were largest close to the loading point. The wave-like pattern in the lower part of the pontic indicates that these were the locations of high tensile stress concentrations (indicated by the arrows). From the wave-like pattern it could be also seen that the tensile stress concentration on the right side was larger than on the left side. Therefore, the fracture probably started from the right side of the connector.

3.2. Acoustic emission (AE)

Acoustic emissions (AE) are the stress waves induced by the sudden internal stress redistribution of the materials caused by the change in the internal structure. In the R-T2D code, it was assumed that the strain energies released by damaged elements were all in the form of AE. The AE energy was mathematically calculated in the R-T2D code by Eq. (2):
\[ \Delta e_i = e_{ic} - e_{ir} \]
where \( e_{ic} \) is the elemental strain energy of element \( i \) before failure, \( e_{ir} \) is its strain energy after failure. The elemental strain

![Fig. 2](image)

Fig. 2 – Simulated quasi-photoelastic stress fringe pattern caused by loading in the central axial plane of the framework. The fringe pattern shows the stress distribution for maximum principal stress (A) and minimum principal stress (B). Arrows mark the highest tensile stress concentration.
energy depends on the elastic modulus of the element, the stresses before and after failure, and the volume of the element. The failure of one element was represented by a circle with the centre located in the centre of the element. The radius of the circle represented the magnitude of the energy release from this element. That is, a larger circle represented a failed element with higher elastic energy, making it easier to characterize the heterogeneity of the material.

The calculation of AE could be performed visually in the R-T2D code. The color red represented tensile failure at the current loading level; blue represented the compressive (or shear) failure; whereas black represented the cumulative failures at the previous loading levels. In Fig. 3 one can note that the fracture was initiated at the lower portion of the right connector area near the pontic (Fig. 3A and B), meaning that the connector area near the pontic was the weakest part of the framework. In Fig. 3 one can even see that the breakage occurred in only one single loading level (Fig. 3A to F), which means that even if the load is kept constant the crack continuously propagates until there is complete failure of the framework. This is a typical phenomenon of brittle failure.

3.3. Stress distribution, fracture process and fracture pattern

The crack started at the ninth step and the whole fracture process happened during this step. The chain reaction of the fracture process was calculated through 43 steps within the ninth step. In Fig. 4, the maximum principal stress distribution is presented from A1 to A5, while the minimum principal stress distribution is presented from B1 to B5. As soon as loading started, a high value of maximum principal stress concentration could be observed in the framework near the loading point, whereas lower values of maximum principal stress concentration were observed in the lower portion of the connector areas near the abutments (Fig. 4A1). In Fig. 4B1, high minimum principal stresses were observed in the lower portion of the connector area near the pontic. In Fig. 4B2–B5, the cracks sprout first at the lower edge of the right side of the connector near the pontic. As mentioned earlier, the minimum principal stress described tensile stress in the current study. Therefore, the cracking mechanism of the framework was tensile failure. When the development of the crack continued from Fig. 4A2 to A4, the concentration areas of the maximum principal stresses on the right side of the connector area gradually disappeared. Finally, in Fig. 4A5, the maximum principal stress concentration areas
shifted mostly to the left side of the connector area near the left abutment. The fracture process of the framework was completed by a crack starting at the lower portion of the connector area near the pontic and extended to the loading point. From this analysis one could see that the fracture pattern is determined by the stress distribution in the framework and the stress distribution varied continuously in the fracture process.
Fig. 5 – A representative view of a line drawn in the 3 U Y–TZP FPD framework along the fracture line from Q1 to Q2 (A). The stress–element curve shows the shear, the maximum and minimum principal stresses before fracture (B) along the fracture line. The Y-axis represents the values of stress in MPa and the X-axis indicates the element number (B).

3.4. Stress distribution along a line

In the R-T2D code, stress distribution corresponding to any loading step along any line in the structure can be calculated, giving a more detailed description of the stress distribution in a particular area in a selected step. A line could be set anywhere in the framework after the fracture calculation is finished, and the code can automatically calculate the shear stress, maximum principal stress and minimum principal stress for each element in the whole fracture process along the line. In the present study, a line was drawn along the fracture from the upper portion of the framework (Q1) towards the lower portion (Q2) (Fig. 5A) and the stress distribution of the loaded framework before fracture was calculated. There were in total 77 elements along the line and the elements were numbered from 1 to 77 from Q1 to Q2. In Fig. 5B, the stress–element curve for the ninth step before fracture is presented. One could note that the highest values of maximum principal stress were observed from the first to the third element. The magnitude of the shear stress is smaller than the maximum compressive stress. A low value for minimum principal stress can be seen in that part of the framework. Eventually, the maximum principal stress becomes lower and lower, especially for the lower portion of the framework but the tensile stress (minimum principal stress) becomes larger and larger and reaches its maximum at the edge element. Once the maximum tensile stress reaches the tensile strength of this element the element will fail due to tension. The crack started from the place where the largest tensile stress was located, which was in the right lower portion of the framework (Fig. 5B).

4. Discussion

In the current study, the fracture process of a one-layer 3 U FPD framework was studied. Usually, all-ceramic FPDs have two layers, a central part made of a reinforced core material and an outer part made of a feldspathic or glass ceramic. The strength of bilayered core/porcelain specimens can be affected by the properties of the veneering porcelain and the core/veneer thickness ratio [8,19,20]. Abutment conditions, such as the material of the abutments and whether the abutments are fixed or mobile, can also affect the strength and stress distribution of all-ceramic FPDs [21], and in reality, teeth consist of enamel, dentin and a pulp chamber and are surrounded with periodontal ligaments and bone. However, in order to ensure that the numerical set-up is closely related to a previous laboratory study and, thus, would facilitate comparisons between the studies, a one-layer ceramic framework was selected in the present study and the parameters of fixed stainless steel abutments were used.

Comparison between the present numerical simulation and the previous laboratory test [6] revealed that the fracture patterns were in close agreement with each other. In both the numerical study and the laboratory test [6], fractures were observed only in the ceramic; no fracture could be observed in the stainless steel abutments. Fig. 6A shows a representative example of a HIPed Y–TZP FPD framework fractured through one of the connectors and Fig. 6B shows the result of numerical simulation using the R-T2D code. The location of the fracture in both studies was between one of the connectors and diagonally through the loading point. One important difference between the studies was that in the present numerical simulation the whole fracture process could be followed step-by-step, whereas in the laboratory test [6] the fracture occurred very quickly, in a moment. Since fracture of dental all-ceramic FPDs usually happens fast and often cannot be followed in laboratory studies or in vivo, the current numerical simulation provided interesting information in that the fracture process could be visualized from the beginning to the end.

In other tests [11,22] dealing with failure analyses of three-unit all-ceramic FPDs, it has been stated that the initiation of fractures was in the gingival embrasure [22], and using finite element analysis it was shown that the maximum tensile stresses occurred at the lower side of the connectors [11,22]. These results are in agreement with the findings in the current numerical simulation, in which the cracks always started at the lower boundary of the framework. That is, where the highest concentration of tensile stresses was located (Fig. 4B2).
Thus, based on those findings the weakness in three-unit all-ceramic frameworks subjected to the current loading condition seems to be in the lower boundary. In addition, superficial cracks and defects may have a negative impact on the strength of all-ceramic frameworks, especially in the lower boundary. Since grinding and polishing can reduce superficial cracks [23], it should be possible to make all-ceramic constructions stronger, more durable and longer lasting by strengthening and smoothing the lower boundary. This is a supposition that needs to be further analyzed.

In order to mimic the loading conditions more closely to when frameworks have been seated manually and, thus, not exactly symmetrically, the load in the numerical simulation was applied not exactly in the middle of the pontic, but shifted five elements towards one of the connectors. For the numerical simulation a more balanced and almost exactly symmetrical loading could be achieved compared to loading in the laboratory test. In a preparatory study, when the simulated load was placed as exactly as possible in the middle of the framework the cracks started simultaneously from both sides of the connector areas at the lower boundary. In addition, in order to evaluate the effect of the pontic design on the fracture pattern, a symmetrical model with a longer and flatter pontic than that in the laboratory study was evaluated in another preparatory study using the R-T2D code. The results of the change in the laboratory study was evaluated in another preparatory study. For the numerical simulation a more balanced and almost exactly symmetrical loading could be achieved compared to loading in the laboratory test. In a preparatory study, when the simulated load was placed as exactly as possible in the middle of the framework the cracks started simultaneously from both sides of the connector areas at the lower boundary. In addition, in order to evaluate the effect of the pontic design on the fracture pattern, a symmetrical model with a longer and flatter pontic than that in the laboratory study was evaluated in another preparatory study using the R-T2D code. The results of the change in the framework design revealed that the fracture still started at the lower boundary of the framework, but the crack initiation was not in the connection area. Instead, the cracks started at a number of locations in the middle of the lower portion of the pontic. In light of the above findings, a longer and flatter pontic would appear to make the tensile stress concentration spread out over a larger area and seemingly not affect the connectors in the same way as in frameworks with a pontic design similar to that used in the laboratory test (Fig. 6A). These indicate that the fracture pattern of the framework can be influenced by modifying the pontic symmetry and/or shape.

In the present study, the stainless steel was assumed to be a homogeneous material which stands compressive strength as much as tensile strength. Therefore, the value for the homogeneity index/Weibull’s modulus was set at 100. The value of the tensile strength for Y–TZP could not be found in the existing literature, but according to the manufacturer’s information the flexural strength of Y–TZP was said to be almost similar to the value for the tensile strength, which they had verified in earlier finite element analysis. Therefore, the data for tensile strength in Table 1 was taken from the material’s uniaxial flexural strength given in a previous paper [15]. The R-T2D code takes material heterogeneity into consideration. The highest tensile stress just before element failure, shown in the stress–element curve in Fig. 5B, was around 800 MPa. That is, less than the value given for tensile strength in Table 1. This is probably because the heterogeneity of the material observes Weibull’s distribution law. The local strength for some elements can be higher than the strength given in Table 1, but for some others it can be lower and Table 1 may only show an average value. In addition, the stress value depends on the material properties, the load at that step as well as the area to carry the load. The smaller the area, the higher the stress would be for the same magnitude of load and the same material. Higher shear stress would be expected if higher compressive stress were caused by loading. Accordingly, shear stress has the possibility of reaching the strength criterion and therefore, breakage mechanisms can be changed into compression (or shear) failure. However, this was not the case in the present study.

In the current study, the mechanical loading was solely static and carried out on immobile abutments and the simulated mechanical loading was simplified to a two-dimensional plane strain condition. This kind of modeling should have captured the basic failure mechanisms dealing with 3 U Y–TZP FPD frameworks and R-T2D code seems to be able to serve as a complement to other tests and clinical observations in studying fracture processes in all-ceramic frameworks under simulated static loadings. However, in the oral cavity, the load is more likely to be a three-dimensional problem and there are several factors influencing the fracture process of all-ceramic FPDs. Further laboratory and numerical tests are, therefore, needed in order to elucidate the effects of different abutments, loading conditions, cementations, ceramic layers, FPD shapes and dimensions on fracture processes in all-ceramic FPDs.

5. Conclusions

Within the limitation of this study the following conclusions were drawn:

![Fig. 6 – Fracture pattern of a 3 U Y–TZP FPD framework in a previous laboratory test using a universal testing machine (A) and the results of the numerical simulation using the R-T2D code (B). The arrows in (A) indicate the fracture line. The grey scale in (B) represents the distribution of Young's modulus: the brighter the color, the higher is the value of Young's modulus. The black line indicates the fracture.](Image 1)
1. There was a fairly good correlation between the fracture process results obtained using the R-T2D code and a previous laboratory study of 3 U Y–TZP FPD frameworks.
2. Numerical analysis using the R-T2D code revealed that the fracture mechanism of the 3 U Y–TZP FPD framework was tensile failure.
3. Cracks in 3 U Y–TZP FPD framework start at the lower boundary.
4. Based on the findings in the current study, the R-T2D code seems to be able to serve as complement to other tests and clinical observations for assessing fracture processes in three-unit all-ceramic frameworks.

Acknowledgements
This work was supported by the Kempe Foundation. The authors are also grateful to Prof. C.A. Tang, Dalian University of Technology, Dalian, China for his help.

REFERENCES