Polymerization and loading stress distribution in adhesive resin-based composite class II restorations
Ausiello, P.

Citation for published version (APA):
Ausiello, P. (2002). Polymerization and loading stress distribution in adhesive resin-based composite class II restorations
3D finite element analyses of cusp movements in a human upper premolar, restored with adhesive resin-based composites

Introduction

Resin-based materials are being employed to a large extent in contemporary restorative dentistry. A prominent advantage is the possibility of bonding the restoration to tooth structure. Significant disadvantages of these materials are the polymerization contraction and the mismatch in mechanical characteristics with tooth structures (Davidson, 2000). During hardening, polymerization shrinkage stresses the adhesively placed restoration, while the mechanical mismatch leads to interfacial stress concentrations during functioning (mastication, temperature fluctuations). Modifying both material characteristics by adding inorganic filler particles to the resin, results in shrinkage reduction and strength and stiffness (E-modulus) increase. Unfortunately, increase of modulus is associated with higher polymerization shrinkage stresses. Stress control in resin-based composite restorations is a major problem in operative dentistry.

Clinical studies have shown that fractures in teeth with Class II MOD restorations are mainly related to insufficient residual hard tooth structure, restorative procedures and material selection (Wendt et al., 1987). Occlusal loading of teeth, adhesively restored with a variety of materials (amalgam, resin-based composites and combinations of

glass-ionomers with resin-based composites), did not lead to equal fracture resistance values nor to comparable fracture patterns (Ausiello et al., 1997). Resin-based composite restored teeth often show mesial-distal crown fractures with the composite material fracture located at the critical tooth-resin interface. This behaviour has been attributed to the stresses arising from the polymerization shrinkage (Davidson and De Gee, 1984), which results in stress build-up in the restored tooth and cusp displacements (Pearson and Hegarty, 1987). Repeatedly functional loading will cause fatiguing of the restored tooth and will ultimately result in failure. Ausiello et al. (1999) investigated \textit{in vitro} the performance of adhesively restored large and deep cavities in maxillary premolars under cyclic loading. That fatiguing process showed that, when polymerization shrinkage pre-stressing conditions are present, the adhesive restoration-cavity wall interface initially debonds at the weakest bonded areas between the dentine and the restorative material. As reported in several previous studies (Koike et al., 1990), polymerization shrinkage stress of adhesive composite restorations depends on several factors such as restoration size, cavity shape, incremental or bulk placement technique, water sorption, composite creep and cusp movements (Feilzer et al., 1990). Moreover, it has been shown, that increased flexibility (reduced E-modulus) of the restoration reduces the polymerization stress (Kemp-Scholte and Davidson, 1990). Suliman et al. (1993) investigated \textit{in vitro} the cusps deflection in adhesively restored natural teeth under different polymerization compensating effects, by using a microscope with a micrometer. According to that study, cusps deflection differences depend on cavity size, wet and dry storage conditions and filling material rigidity. Apparently there are a variety of factors that are simultaneously active. Some enlarge one another, whilst other mechanisms counteract one another. Proper understanding of the physical phenomena occurring in the adhesively restored tooth during the placement and setting of the materials and throughout its functioning.
is crucial in the determination of the proper placement procedures and material selection. Structural analyses of the integral process, may offer an essential tool for the assessment and reduction of fracture risks, ensuring optimal performance in selection of the restorative material combinations and material application protocols. Since stress distribution investigation of teeth, in particular after restoration, is very complicated due to the complex geometry, 3D finite element analysis (FEA) might be a powerful tool to visualize the problems.

The aim of this study was to provide FEA engineering tools for the understanding of the influence of the shrinkage characteristics and composite rigidity on the amount of cusps displacement and localization of critical sites in a given restored tooth under (over) functional loading. Moreover, validation of the proposed procedure of the finite element model (FEM) was investigated by a comparison between the calculated theory and the mechanical experiment data.

Materials and Methods

3D finite element approach consists in dividing a geometric model into a finite number of elements in which the variables of interest are approximated with some mathematical functions. Biomedical applications of this method are already known in some fields of medicine (Apicella et al., 1994; Dalstra et al., 1995; Apicella et al., 1998).

In our study, a 3D FEM model of a first upper premolar was realized. The first step was the solid model generation in which the shapes of dentine, pulp and enamel were obtained. The space for the Class II MOD cavity preparation was also defined. Then the solid model was filled with elements, creating the mesh, and the material properties of dentine and enamel were assigned to the elements, which filled the corresponding regions. Thus the sound tooth FEM model
was obtained. Subsequently, the FEM model of the restored tooth was realized by changing element material properties in the central zone of cavity preparation. At the end loads and geometric conditions were applied.

**Tooth solid model generation** - Tooth solid model was generated using literature data on the tooth morphology for the definition of the dentine and enamel volumes (Wheeler, 1974) and a plaster model (Thanaka model, Japan, 1978) for the external shape definition. Crown and roots were constructed in two different phases and assembled afterwards. The crown was realized by digitising an upper premolar tooth plaster model on the scale of one to five by a Cyberware laser scanner. Over two hundred profiles were generated at 0.33 mm increments by laser scanning in two different directions: a vertical and a horizontal one. Among all the profiles, only 34 were collected, 17 vertical and 17 horizontal at 2 mm increments, and were assembled in a 3D wire-frame structure (Fig. 1) by means of a 3D CAD (Autocad 12, Autodesk, Inc., Neuchatel, Swiss, 1992). The

![Crown wire frame and cut in cervical area](image1)
![Tooth dentine](image2)

Fig. 1 - 3D solid upper premolar model and finite element model generation.
wire-frame curves were exported in Pro-Engineer 16.0 (Parametric Technology Co., Waltham, MA, USA, 1994), where a solid model was generated fitting the horizontal and vertical profiles. The model was cut in the cervical area in order to obtain the final crown (Fig. 1).

The roots were modelled by means of their mesial-distal and buccal-lingual representations taken from literature (Braden, 1976). The two representations were scanned and 8 vertical profiles were generated imitating the scanned images. The roots were constructed, fitting the vertical profiles (Fig. 1). The pulp region was obtained in an analogous way and subsequently subtracted from the roots.

The crown and the roots with the pulp chamber were assembled in the final model (Fig. 1). Defining parametric cutting plane, it was also possible to realize easily different cavities and MOD preparations. In Fig. 5, a MOD II cavity is shown with 3.5 mm occlusal width.

**FEM model generation** - The solid model was exported into ANSYS rel. 5.3 (Ansys Inc, Houston, USA, 1994), using the IGES format. The volumes were redefined and meshed with 8-node brick and 4-node tetrahedral elements, resulting in 11165 elements and 7340 node structure. Different material properties were assigned to the elements according to the volume definition. Three different models were realized. The first model (Mod. A) was the sound tooth; the second model (Mod. B) was a restored tooth with a more rigid composite; the third model was a restored tooth with a less rigid composite (Mod. C). Material properties are listed in Table 1.

Some assumptions were made in order to simplify the calculations. Absolute bonding was considered among enamel, dentine and composite. The pulp chamber was modelled as a void because of its negligible stiffness and strength. Despite of their intrinsic anisotropic nature, dentine and enamel can be assumed homogeneous and isotropic (Darendeliler et al., 1998; Versluis et al., 1996) because their anisotropy belongs to a microscopic scale whereas the tooth model is
Table 1

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus [GPa]</th>
<th>Poisson ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dentine</td>
<td>18 (a)</td>
<td>0.23 (a)</td>
</tr>
<tr>
<td>Enamel</td>
<td>48 (b)</td>
<td>0.3 (b)</td>
</tr>
<tr>
<td>More rigid composite (B)</td>
<td>25</td>
<td>0.3</td>
</tr>
<tr>
<td>Less rigid composite (C)</td>
<td>12.5</td>
<td>0.3</td>
</tr>
</tbody>
</table>

(a) Wheeler, 1974; (b) Versluys, 1996.

macroscopic. Furthermore all materials were considered elastic throughout the entire deformation, which is a reasonable assumption for brittle materials in non-failure conditions (Rees et al., 1995)

**Model experimental validation** - In order to validate the FEM model, a compression test was performed on a Class II MOD restored human upper premolar. The resin-based composite Prodigy (Kerr, USA), with a Young’s modulus of 12.5 GPa and polymerization shrinkage of 2.73 ± 0.31% by volume in combination with Optibond FL adhesive (Kerr, USA) was used for the restoration. The tooth was inserted up to the cementum-enamel junction in a steel cylindrical ring with the apical root area in contact with the steel ring floor. Subsequently it was filled with resin composite material, so it was possible to record only the material deformation within the tooth. The cylinder was clamped to the test machine and the load was applied vertically by means of a 6 mm diameter steel cylinder with the axis parallel to the tooth axis in order to simulate one main important force which develops during occlusion (Mc Neil, 1997). A 1 mm/min constant rate was imposed to the loading cylinder and the vertical displacement and the axial load were acquired until the restored tooth fractured. The same test was simulated using the FE analysis and the results
Fig. 2 - Occlusal vertical load and comparison between experimental and numerical data.

compared (experimental curve in Fig. 2). The comparison test loading cylinder was modelled with the 3D elastic beams. The beams have one end in common and the other one on a cusp. End rotations were not constrained. The common end was displaced in the central position of the loading cylinder section in the experimental test. The load was applied on the tooth at two points (Fig. 2) through the beams elements (blue lines) on the cusps (red areas). The resulting force F crossing the central position of the loading cylinder section was 400 (N) (Fig. 2, red arrow). The beam elastic properties were thought to be infinitely rigid compared to the tooth. Resin support was not modelled as it was considered to be as rigid as the loading system. Other studies (Darendeler et al., 1998) also assumed the supporting alveolar bone to be rigid.

Numerical simulations - Three different situations were simulated: a compression test with a 400 N occlusal load, composite polymeriza-
tion shrinkage and the combination of occlusal loading and shrinkage. In these simulations, the tooth model had to be constrained apically to the enamel-dentine junction. This fact permitted to eliminate the lower part of the mesh reducing the model size to 7894 elements and 5812 nodes.

Numerical results were obtained for the three models (A, B and C). Particularly the vertical displacement of the load application point in the model C can be used as the numerical datum for the model experimental validation.

For the evaluation of the stresses arising from the composite polymerization shrinkage, a volumetric contraction was applied to the composite. The shrinkage data required for this study were obtained by using a linometer (De Gee et al., 1993). The shrinkage which determines cusp deflections, however, cannot be directly related to the amount measured in the linometer free shrinkage tests. This is due to the viscoelastic relaxation occurring in the material during the passage

**On-off experimental test**

![Low intensity](image)

![High intensity](image)

**On-off test FEM simulation**

![FEM simulation](image)

**FIG. 3 - On-off experimental test with FEM simulation.**
from a viscous fluid into an elastic solid. It has to be noted that when a composite is adhesively placed in the tooth cavity, only a part of the shrinkage produces cusp deflections and a part relaxes. In this paper an on-off test is described: composite discs were polymerized on a glass substrate (Fig. 3). In that study, the composite shrinkage stresses should overcome the silica glass substrate strength at about 20% of the value measured in the linometer test; nonetheless, fracture was observed in a few cases only when total shrinkage was attained. This datum was arbitrarily used as reference in this study. A 0.54% constrained volume shrinkage (2.7% free volumetric shrinkage) was assumed to occur during the inside cavity resin-based composite polymerization.

**Results**

All the analyses performed were linear static. The comparison between the experimental curve and the numerically determined one is shown in Fig. 2. Fig. 4 shows the Von Mises equivalent stress which

![Von Mises stress distribution](image)

**Fig. 4.** - Von Mises stress distribution due to vertical loading (up) and displacement distribution (down).
was evaluated for the three models under vertical (over)-functional loading as well as total displacement.

In Fig. 5 the path traces are indicated with the two yellow small circular points. Both the paths are normal to the tooth axis, mesial-

distally directed, little above the pulp chamber and near the interface of model B and C between composite and natural tooth. One path is on the side of the composite restoration; the other path is on the natural tooth side. Von Mises equivalent stress was evaluated along these paths (Figs. 6 and 7).

![Fig. 5 - Paths defined in tooth.](image)

![Stress path in cusp](image)

**Fig. 6 - Von Mises stress path in cusp (natural tooth).**
A Von Mises equivalent stress map was also extracted for the model B and C under shrinkage condition (Fig. 8). Two more paths were defined on these models, following the buccal and lingual wall profiles of the tooth (Fig. 9). Along these paths normal displacement to the tooth axis was determined. In Fig. 9 the cusp displacements are plotted. Lingual cusp displacement is considered negative towards the
centre of the tooth, while buccal cusp displacement is considered positive towards the centre of the tooth.

In Fig. 10 the Von Mises equivalent stress map shows models B and C under the vertical loading and shrinkage combination and model A under vertical loading only.

Fig. 10 - Von Mises stress distribution due to vertical loading for model A. Shrinkage and vertical loading for models B and C.
Discussion

Endodontically treated teeth are easily subject to fracture as a result of the modification of their natural rigidity and to pre-stressing, generated by the restorative procedures. Conventional conservative class II MOD composite restorations may exhibit micro-cracking and interfacial failure due to internal pre-stressing from altered cusp movements as a result of resin-based composite polymerization contraction and occlusal loading. The study of the effects of restoration techniques on endodontically treated teeth usually encounters all the negative influences associated with the use of natural tooth samples. Teeth, in fact, differ from one another for obvious reasons, patient age, sex, etc. That is the reason for large standard deviations of the determined means values. Therefore, different laboratory test results are difficult to compare. Consequently it is of paramount importance to firstly exclude the anatomical sample differences and statistical variability. In the present study, the mechanical behaviour of a bicuspid, subjected to polymerization contraction and occlusal loading, has been investigated by means of a computer analysis and numerical results were validated by laboratory experimental data. FEM provides a powerful tool for analysing the mechanical behaviour of complex structures. For this reason in the last two decades FEM has been widely used in studying biological systems. In fact, a mathematical model permits to evaluate natural systems response under various load or geometric conditions apart from the high dispersion, which characterizes experimental data. Obviously, FEM models need an experimental validation, which is purposely prepared and definitive. If possible, suitable laboratory data for validation can also be collected from literature in order to minimize time and costs. After validation, the model can be extensively used for a wide range of studies. Only significant changes in the FEM model require a new experimental validation.
The system analysis procedures proposed in this study create a feedback mechanism that helps in closing the structures and material design loop with engineering prediction of application stresses localization and build-up coupled with a structural performance. In the last two decades many works have shown how the 2D finite element analysis applied to dental mechanics has become a popular numerical method to investigate the critical aspects related to stress distribution in tooth (Sakaguchi et al., 1991) and in dental restoratives (Farah and Craig, 1974) or to evaluate the stress relief of shrinking resin-based materials (Versluis, 1996).

The use of a more detailed 3D dental model to test the influence of elastic and shrinking properties of resin-based restoring material commonly used in operative dentistry (such as light curing resin based composite), seems to be of extreme interest to understand critical problems related to the restorative material choice and optimal application procedures definition.

We focused our interest in the investigation of the mechanical behaviour of a critical system such as the first upper premolar restored by a class II MOD cavity preparation (where a consistent amount of dentine and enamel tissue is lost and the integrity of the structure is seriously altered).

Methods for geometric data acquisition, geometric model creation and modification (parametric modelling) and finite element model generation through CAD ambient are presented and discussed.

The FEM prediction is in good agreement with the experimental data, as shown in Fig. 2. Comparison underlines the great accuracy for the tooth model with respect to a very complex simulation as a compressive test of restored tooth. Our numerical data are also in good agreement with experimental ones obtained by Suliman et al. (1993) by means of a microscope with a micrometer stage.

Considering, firstly, the occlusal loading effect alone, the sound tooth exhibits a wider high stress area (red), localized in correspondence
of the occlusal enamel, than the restored teeth (Fig. 4, upper left side). This is due to the rigidity of the enamel. In particular, looking at the models B and C, in which a different composite rigidity was used, wider stress areas were found in the model with the more rigid composite (Mod. B). In fact, a less rigid structure (Mod. C) releases stress in greater deformation. In this way, Fig. 4 shows how the cusp movements increase from model A to C. Even if high stress areas are reduced in Mod. C compared to Mod. B, it does not mean that its mechanical behaviour is the best one. In fact, this reduction occurs in the composite restoration, which is less rigid, and its lower rigidity allows greater cusp movements. So the average stress in the entire structure is lower but the stress values in buccal and lingual cusps are higher. These preliminary considerations, however, do not take into account setting composite shrinkage. Obviously, the best behaviour is relative to the sound tooth model with a very large stress area in the central enamel region and low cusp deflections. This consideration is confirmed by the Von Mises stress path into the natural tooth (Fig. 6). The peaks on the left and on the right side depend on the enamel, which is more rigid than dentine. The curves have the same shape for the three models but there is higher stress for the less rigid structure. It is interesting to note the difference from this Von Mises path and the Von Mises path in composite (Fig. 7). Along the two different paths the sound tooth exhibits an analogous behaviour, with two peaks in enamel. These two peaks disappear for the restored models in which there is a single material: the composite. In Fig. 7, in fact, Mod. B and Mod. C have similar curves but Mod. B exhibits higher values. Mod. C behaves in the central zone as the dentine of the sound tooth. In applying contraction load in models B and C, the high stress areas are concentrated near the composite-tooth interface, where failure can occur (Fig. 8). Even if failure does not occur during composite curing, a pre-stress rises in the structure. Higher modulus restoration exhibits higher stress values. In fact, a less rigid restoration can relax the
applied stress by means of a greater elastic deformation. This effect is also shown in Fig. 9 where the displacements normal to the tooth axis are plotted. The less rigid composite exhibits a greater elastic deformation that is transferred to a lower deformation of the cusps. Higher stress release depends also on viscous flow of the composite during curing. But this effect was not directly taken into account in these models. Simply, the contraction load input in the simulations was significantly reduced compared to the free contraction.

At the end, occlusal loading and shrinkage were combined for models B and C. A Von Mises stress map is plotted in Fig. 10 together with that of the sound tooth under occlusal loading. Mod. B and Mod. C exhibit a similar behaviour, even if Mod. B has a higher stress in the upper part of the restoration whereas Mod. C has a higher stress in the cusps. However, this effect is greatly reduced compared to the single occlusal loading case. The important difference is between these two models and the sound tooth because they show high stress values at the composite-tooth interface, in a zone where the sound tooth exhibits a low stress.

This study was designed to investigate the correlation between resin composite elastic properties and restored tooth stiffness. The findings indicate that more rigid composites lead to lower cusp movements under occlusal loading but exhibit a higher pre-loading effect. A good composite for restoration has to balance the two opposite effects. In this way a low pre-load on the cusps can be accepted in order to reach sufficient restoration rigidity.

Conclusions

FEA simulations supply information about the mechanical behaviour of sound and restored tooth. The best restoration is the one which allows restored tooth to react to external loads as the sound
tooth does. That is why the composite has to store a great quantity of energy, limiting cusp movements. Furthermore it does not have to bend cusps as a consequence of its curing, otherwise the pre-loading effect can be disastrous. The problem is that rigid composites have a low elastic release. In this way the solution is in balancing the two effects, accepting a low pre-loading effect in order to obtain sufficient rigidity.

A new point of view can be introduced considering two aspects not contemplated in the present model. On one hand there is the viscous flow that can generate a great stress release during composite curing. But viscous flow needs time and actually the light curing composite kinetics are too fast for this. On the other hand there is the adhesive interface, which can limit the cusp pre-loading effect with its elastic deformation. By changing the adhesive elastic modulus or simply changing its thickness, it is possible to obtain a different rigidity of the interface region. In this direction we plan to develop the present study, making use again of FEA simulations, which prove to be an efficient tool in complex structure analysis.