Inlay Fixed Partial Denture Framework 3-D Structural Integrity Validation Using COMSOL Multiphysics 3.5a

Timo M. R. Alho*
Department of Electrical Engineering and Automation, University of Vaasa, Finland.
*Corresponding author: University of Vaasa, Wolffintie 34, FI-65101 Vaasa, Finland; timalh@uwasa.fi

Abstract: Manual manufacturing of inlay fixed partial denture frameworks by metal casting can take hours of dental practitioners work time. This paper introduces 3-D simulations of pre-manufactured inlay fixed partial denture framework assembled from laser cut sheet metal parts. The study gives a good estimation of how well the frameworks can withstand strong human occlusion forces and masticatory cycle. The simulated forces corresponds to average maximum biting force of an adult young male in static analysis and in time depended simulations the acting forces are scaled down to mimic the masticatory cycle with changing force amplitude.

Keywords: Framework, inlay fixed partial denture, structural integrity, pre-manufactured

1. Introduction

Making dental prosthesis, like the one in Figure 1, is a delicate and time consuming work. One very common way to manufacture inlay fixed partial denture (IFPD) frameworks is metal casting. It allows manufacturing a framework with virtually any shape, but the downside is that it can take hours to manufacture even one framework. Although there are many milling machines combined with 3-D scanner on the market which can make the prostheses automatically, but very few dental practitioners can afford to buy these machines. For this reason most of the prostheses are still made by hand in a laboratory.

Figure 1. Inlay fixed partial denture attached to a cast jaw.

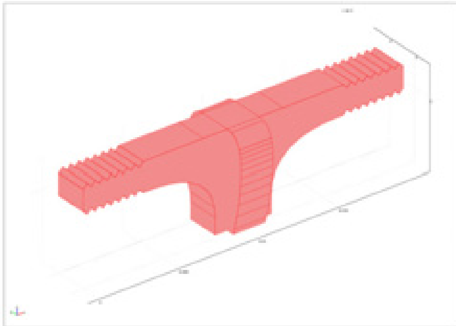

Figure 2. 3-D model of inlay fixed partial denture framework.

This paper introduces 3-D structural mechanics simulations conducted with COMSOL Multiphysics 3.5a to validate the structural integrity of a pre-manufactured IFPD framework. The 3-D model of the framework is shown in Figure 2.

IFPDs are minimally invasive way to replace a missing tooth. [1] The attachment of the prosthesis is done using attachment wings. Depending on the size of the abutment teeth the wings are either embedded into the teeth (Fig. 1) or attached to their back surface.

The proposed framework is designed to replace a single missing tooth in parts of the molar and pre-molar areas. In this area the teeth have sufficient size to enable the kind of embedded attachment as in Figure 1 and this makes the framework's manufacture process much simpler. The framework is laser cut from 2 mm thick sheet metal in 2 parts which are welded together.

These frameworks cannot be used in the incisor area as smaller abutment teeth mean that the attachment would have to be made to the back surface of the teeth. In the cases these frameworks can be used they save hours of the dental practitioners work time, as it takes only minutes to modify the framework using a drill compared to hours of work time needed for metal casting process.

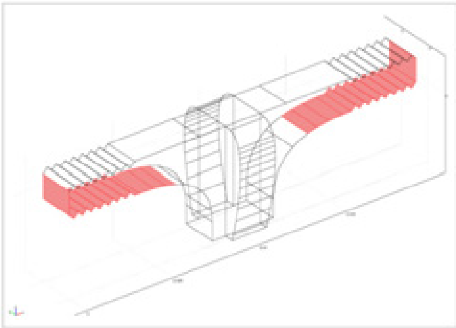

Figure 3. The area from where framework is attached to the simulation environment.

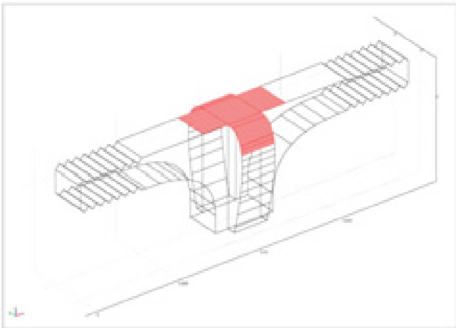

Figure 4. Force implementation area on the framework, the surface area $A \approx 1.73 \cdot 10^{-5}$ m^2.

The mean maximum occlusion force of a young adult male is 287 N and 847 N in incisor and molar areas respectively. [2] The shape of the attachment wings in the framework makes it suitable only in replacing a missing tooth from second pre-molar to second molar where the teeth have sufficient size but also the occlusion forces are the strongest.

2. Use of COMSOL Multiphysics

The simulations conducted with COMSOL Multiphysics vary from simple occlusion simulation with static unidirectional forces to more complicated masticatory cycle simulation with varying force amplitudes and directions.

The framework model is attached to the simulation environment from its attachment wings (Fig. 3) so that there is approximately 6 mm gab between the attachments. This setup mimics a situation where the wings have immensely hard foundation under them and they

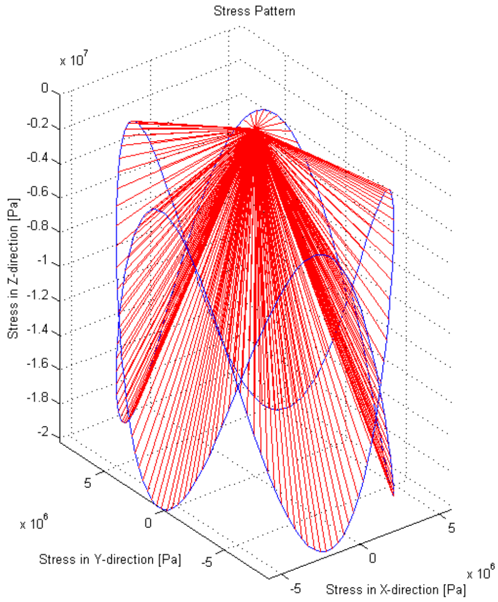

Figure 5. Blue curve represents the pattern of the stress inflected on the framework by the acting forces and red lines represent the stress vectors used in the simulations.

are attached to it by unbreakable cement. This simplification was made to reduce the simulation environment to consist only the framework itself.

The setup causes that the framework model will not be able to move as it would in real situation and the stresses inflicted on it are higher. But as the goal for this study is to verify that will the framework withstand the stresses that occlusion inflicts on it, the higher stresses in the simulations will only give more confirmation if the framework withstands them.

The acting forces in the simulations are implemented to a 5 mm long area in x-direction in the top center part of the framework model to mimic the occlusion forces inflicted on the prosthesis (Fig. 4). In the static simulations the acting force is 847 N which as previously mentioned is the mean maximum occlusion force of young adult male in molar region. When simulating masticatory cycles the acting forces are designed to change in a pattern which takes into account forces inflicted from multiple directions (Fig. 5), the mathematical representation is shown in (1). The acting force

$$\begin{cases} A \approx 1.73 \cdot 10^{-5} \ [\mathrm{m}^2] \\ x = \dfrac{100}{A}\cos(t) \ [\mathrm{Pa}] \\ y = \dfrac{150}{A}\sin(t) \ [\mathrm{Pa}] \\ z = -\dfrac{200}{A} + \dfrac{150}{A}\sin\left(\dfrac{4\pi t}{5}\right) \ [\mathrm{Pa}] \end{cases} \quad (1)$$

$$\begin{bmatrix} \alpha \\ \beta \end{bmatrix} = \begin{bmatrix} \dfrac{4\pi f_1 f_2 (f_1\zeta_2 - f_2\zeta_1)}{f_1^2 - f_2^2} \\ \dfrac{f_1\zeta_1 - f_2\zeta_2}{(f_1^2 - f_2^2)\pi} \end{bmatrix} \quad (2)$$

amplitudes in the masticatory cycle simulations vary between 111.9 and 379.3 N.

The dynamic transformations in time depended simulations were modeled using Rayleigh damping. The Rayleigh damping parameters α and β were calculated using formula (2), where f_1, f_2 and ζ_1, ζ_2 are two of the frameworks' eigenfrequencies and damping coefficients respectively. The values used as damping coefficients were 0.1 and 0.2.

In time depended simulations the modeled masticatory cycle lasts for 12.5 seconds and time steps are taken every 0.05 seconds. This equals to 250 time steps in the time depended simulations.

The density of the mesh in the simulations was compressed as dense as possible to get accurate results. The mesh used is shown in Figure 6 and Table 1 shows some statistics of the mesh.

The material used for the framework was Ti-6Al-4V titanium from COMSOL's own material library with the heat coefficient T = $8.5 \cdot 10^{-6}$ 1/K.

3. Results

Results in both static and time depended simulations are promising as the framework is showing no signs of yielding under the stresses.

Table 1. Statistics of the mesh used in the simulations.

Name	Value
Degrees of Freedom	487,602
Number of Elements	112,129

The von Mises stress values from the simulation results were compared to the yielding stress of titanium which is approximately 920 MPa. [3]

As expected, the stresses culminate in the point where the framework's attachment to the simulation environment begins. In static simulations the von Mises stresses do not exceed the yielding stress of titanium even as, due to the simplifications in the simulation environment, the stresses are higher than they would be in real situation (Fig. 7). Also as expected the maximum bending is unnaturally low, 12.3 µm, due to the simplifications.

In time depended simulations the displacements were also unrealistically small but also in these simulations the von Mises stress does not exceed the yielding stress at any point and there is no noticeable increase in displacement during the simulation. The displacement pattern of the middle point from top of the framework is shown in Figure 8 where maximum displacement is 7.2 µm.

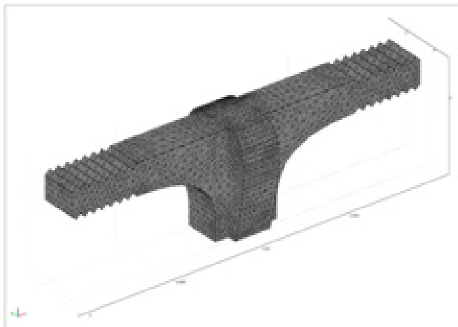

Figure 6. The mesh used in the simulations.

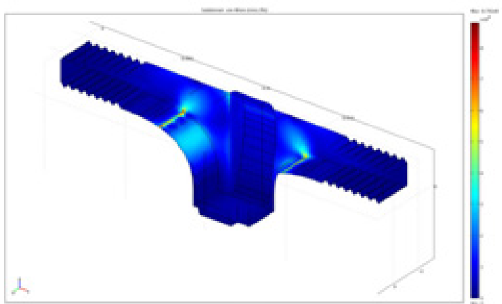

Figure 7. Von Mises stress graph from static simulations using 847 N acting force. Maximum von Mises stress value is 879.2 MPa.

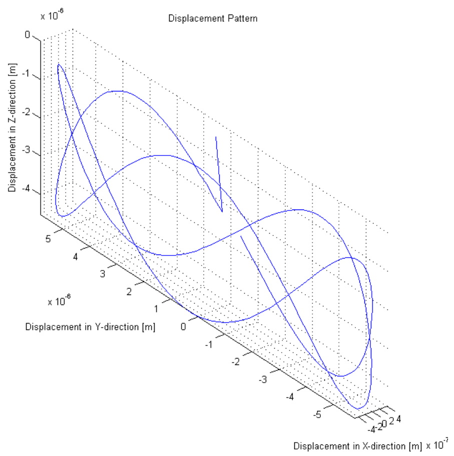

Figure 8. Displacement pattern of the middle point of the framework's top part in the time depended simulations.

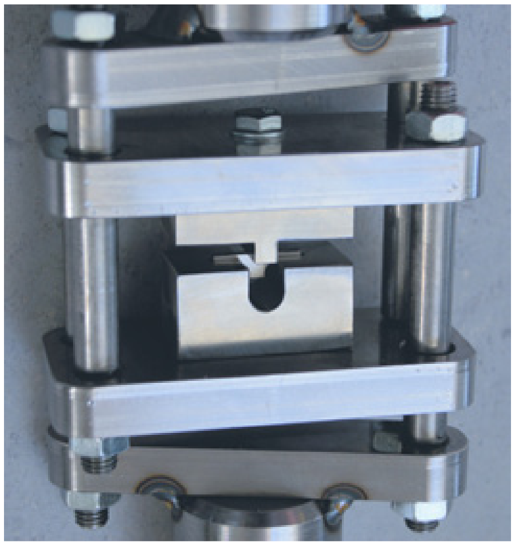

Figure 9. Mechanical testing setup.

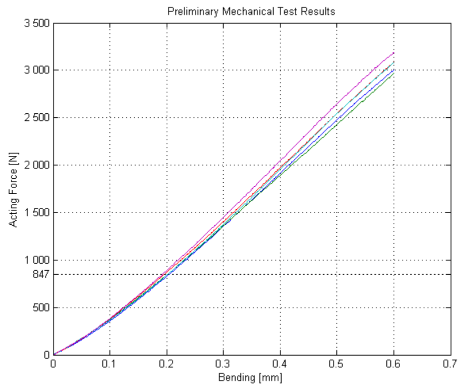

Figure 10. Preliminary mechanical test results using simplified IFPD frameworks.

4. Comparing the Results to Preliminary Mechanical Tests

A small series of preliminary mechanical tests were also performed to simplified versions of the frameworks. These frameworks were cut in one piece and did not have the thicker center part. The tests were simple bending tests where the frameworks were pressed down with ever increasing force. The bending was stopped when the amount of bending reached 0.6 mm.

In the mechanical tests the frameworks are standing on a steel support with 6 mm wide gap in the middle (Fig. 9). The acting force is brought down to a 5 mm wide area in top middle part of the framework. This setting is designed to be as close as possible to the simulations.

In the mechanical tests the frameworks gave in approximately 0.2 mm under 847 N (Fig. 10). As was expected the bending in real situation is much greater than in simulations. This is due to the simplifications in the simulations where for example the attachment wings were not able to move and that way enable more movement.

One way to compensate this in the simulations would be to attach the model to the environment from the two edges which correspond to the steel supports in the mechanical tests. This setup would most likely give more realistic bending results but the simulation results in the vicinity of the edges would be polluted as the edges have no surface area and so in turn also add more unreliability. The best way would be to include the steel supports to the simulations as well.

5. Conclusions and Future Work

From the simulation results it can be concluded that the structural integrity of the presented pre-manufactured IFPD frameworks is sound. The framework model showed no signs of breaking in static or time depended simulations.

Together with the good results from the preliminary mechanical tests this study encourages to continue with further studies.

The study will be continued by optimizing the shape of the framework to minimize the need for dental practitioners to modify it. Also more comprehensive mechanical tests can be done for the frameworks that include more iteration in the bending tests and fatigue tests.

The simulations can be enhanced by including more of the surrounding structures from the abutment teeth and the steel supports when simulating mechanical tests.

A natural next step will also be designing an IFPD framework for the situations where the prosthesis attachment has to be made on the back surface of one or both abutment teeth. This probably can be done using 3-D laser system.

6. References

1. Iglesia-Puig M. A.; Arellano-Cabornero A., Inlay fixed partial denture as a conservative approach for restoring posterior missing teeth: A clinical report. *Journal of Prosthetic Dentistry*, **vol. 89**, p. 443–5, (2003)

2. Waltimo A.; Könönen M., A novel bite force recorder and maximal isometric bite force values for healthy young adults. *Scandinavian Journal of Dental Research*, **vol. 101**, p. 171–5, (1993)

3. Lütjering G; Williams J. C., *Titanium, 2nd edition*, p. 219. Springer, Berlin (2007).

7. Acknowledgements

The author gives his thanks to D.Sc.(Tech.) Harri Hakula for his insights in FEM simulations, Ph.D. L.D.S. Jukka Kentala and C.D.T. Timo Nieminen for their help in dental sciences and Veslatec Oy for manufacturing the framework prototypes for preliminary testing.