Journal of Chemical Health Risks

www.jchr.org

JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

Evaluation of Stress distribution and Fatigue Examination on Implant-supported Fixed Partial denture prosthesis (FPD) with a Zirconia Framework.

[1]Thingujam Debica, [2]Arunoday Kumar, [3]Rajesh S. Nongthombam, [4]Braj Bhushan Mall

[1,2]Assistant Professor, Department of Prosthodontics and crown and bridge, Dental College, RIMS, Imphal, Manipur, India.

[3]Associate Professor, Department of Prosthodontics and crown and bridge, Dental College, RIMS, Imphal, Manipur, India.

[4]Assistant Professor, Department of Oral and Maxillofacial Surgery, Dental College, RIMS, Imphal, Manipur, India.

Corresponding Author: Arunoday Kumar, Email Id: dr.arunodayk@gmail.com

(Received: 27 October 2023 Revised: 22 November Accepted: 26 December)

KEYWORDS

fixed partial denture (FPD), osseointegration, Zirconia, Finite Element Analysis (FEA), finite element model (FEM), yttrium-oxide partially-stabilized (Y-TZP) zirconia.

ABSTRACT:

Objectives: This study aims to find stress distribution and fatigue examination on Implant-Supported Fixed Partial denture prosthesis with a Zirconia framework by finite element analysis (FEA) under different loading conditions. **Materials & Methods**: A 3D finite element model (FEM) is designed. The materials used in the present study are presumed to be homogeneous, linearly elastic, and isotropic. With different angles (0°, 20°, and 45°) and locations, 110 N and 330 N over 0.5 mm^2 areas are applied to the prosthesis, and the equivalent strain distribution of von Mises is studied. Fatigue examination is also done.

Result: At the connector part of the prosthesis, the most extreme pressure is noticed, when the three-unit Zirconia component is loaded midway (110 N load design: 33.1 MPa, 33.2 MPa, and 52 MPa; 330 N load design: 99 MPa, 103.2 MPa, and 156.1 MPa, separately with 0°, 20° and 45° of tendency). The outcomes affirm the weakness in the two connector areas. The assessment of cyclic loading shows a high chance for fatigue, under different loading conditions. No cracks and fatigue are occurring at a load of 110 N. A load of 330 N applied at the centre of the prosthesis doesn't represent any fatigue issue since the load is shared equally by the entire framework. A load of 330 N applied to one of the two connectors, or the two implants, messes exhaustion up. **Conclusion**: FEM analysis of a 3-unit implant-supported Y-TZP FPD, give accurate information about loading conditions for clinical success over time. Fatigue examination results show the structural reliability of the Y-TZP as framework material for 3-unit posterior FPDs.

Introduction.

"Fixed partial dentures" (FPDs) are a promising solution for teeth loss and to regain the patient's chewing activity, health, and aesthetics, and Yttria stabilized tetragonal zirconia (YTZP) FPD is a common practice in prosthetic rehabilitation. For replacing lost teeth or damaged teeth, dental implants are used and this restoration process is called FPD [1]. Dental implants should be biocompatible and have the best mechanical quality because of their intimate contact with the supporting bone (osseointegration) and also because they transfer the occlusal load to the supporting tissues within their physiological resilience levels. Osseointegration is a

Journal of Chemical Health Risks
www.jchr.org
JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

physiological process whereby dental implants are integrated into the bone permanently [2]. Osseo-integrated implants are in direct contact with the bone at the super auxiliary level. The upkeep of this relationship includes persistent redesigning at the bone-implant interface. Peri-implant tissue arrangement and mineralization have to be done carefully as they have to bear mechanical stress at their interface. It is accepted that the strain is initiated in the bone around the implant. Hence, it is essential to see thoroughly how bone reacts to implants under different loading conditions and tissue strains, especially at the implant neck.

Finite element analysis (FEA) is a technique for measuring stresses and strains in complex structures of dentistry. Initially, this method was used only in engineering sectors but due to its various biological advantages and capabilities for modeling and mapping biological structures very well, it has been also adapted in dentistry to find the implant's strength in different fatigue conditions [3]. In this research article, the impact of stress on the substrate of the implant of zirconia FPD is studied and evaluated using the 3-dimensional (3D) FEA, which approximates the stacking states of the structure.

In this study, the latest composite biocompatible material Y-TZP zirconia is used to model the framework of the implant, which improves the dependability and lifetime of muscular implants because of the higher break sturdiness, prevalent, and, mechanical quality of this material [4].

The Y-TZP material has good flexural strength, fracture toughness, and modulus of elasticity of 1000 MPa, 10 MPa$\sqrt{m}$ and, 210 GPa respectively, [4] due to which it can easily offer an all-ceramic bridge even in the posterior region, where higher strength is required. Some literature shows that failures occurred *in vitro* and *in vivo studies*, mainly because of the occurrence of fracture in the connector area of the implant (5, 6-9). The main reason behind these fractures may be a weak model of the framework or chipping of veneering material. Because in chewing activity, the bending forces give tension at the gingival side of the connector area, which leads to cracking and fracture of the framework. This kind of restoration failure may lead to discomfort in the normal activity of the patient or may also create some aesthetic problems. Studies using finite-element analysis demonstrated that during occlusal loading the highest stress within FPDs was located at the gingival side of the connector area (10, 11). Oh et al. (12, 13) demonstrated in a finite element analysis and in an *in vitro* study that connector fracture was initiated at the gingival embrasure and that a larger radius of curvature at the gingival embrasure reduces the concentration of tensile stresses, thus affecting the fracture resistance of the FPD.

The purpose of this study is to use FEA for investigating the effects of various forces like stress and strain in a 3-unit implant-supported Y-TZP FPD. In this research work, a load of 110 N and 330 N loads over 0.5 mm^2 regions with angles 0°, 20° and 45° are applied on the prosthesis and the conveyance of equal von Mises pressure is researched and fatigue investigation and exhaustion examination are completed as well.

Materials And Methods.

Three 2D FEM are created using the software Patran (Version 9)*, comprised of one three-unit, fixed-fixed FPD, and the other two with two-unit cantilever FPDs. The first three-unit model extended from a maxillary canine to a maxillary second premolar. The second model extends from a distal cantilever off the maxillary canine while the third model extends from a mesial cantilever off the maxillary second premolar. In each model, the periodontal ligaments are normalized for a thickness of 0.25 mm with a two-layered supporting cortical bone of 1.5 mm thick and cancellous bone of 1.5 mm thick to make a segment of the maxilla. Retainers of gold alloy cast of 1 mm deep shoulder, are modeled with the connectors and the pontics of standard vertical thickness of 3 mm (10) [14] with a rigid interface without any cement lute.

For the mesh generation, all the FPD models and their supporting structures are fit with eight-noded quadrilateral components with the set boundary conditions for each model. For converging the stresses within 96% accuracy, increased refined meshes are applied to all the models. Table 1 provides the details of the finite element meshes used for FPD models.

Journal of Chemical Health Risks
www.jchr.org
JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

Table 1. *Details of the finite element meshes*

Model	Number of Nodes	Number of elements
Distally cantilevered design	1733	540
Mesially cantilevered design	1729	538
Fixed–fixed design	2422	760

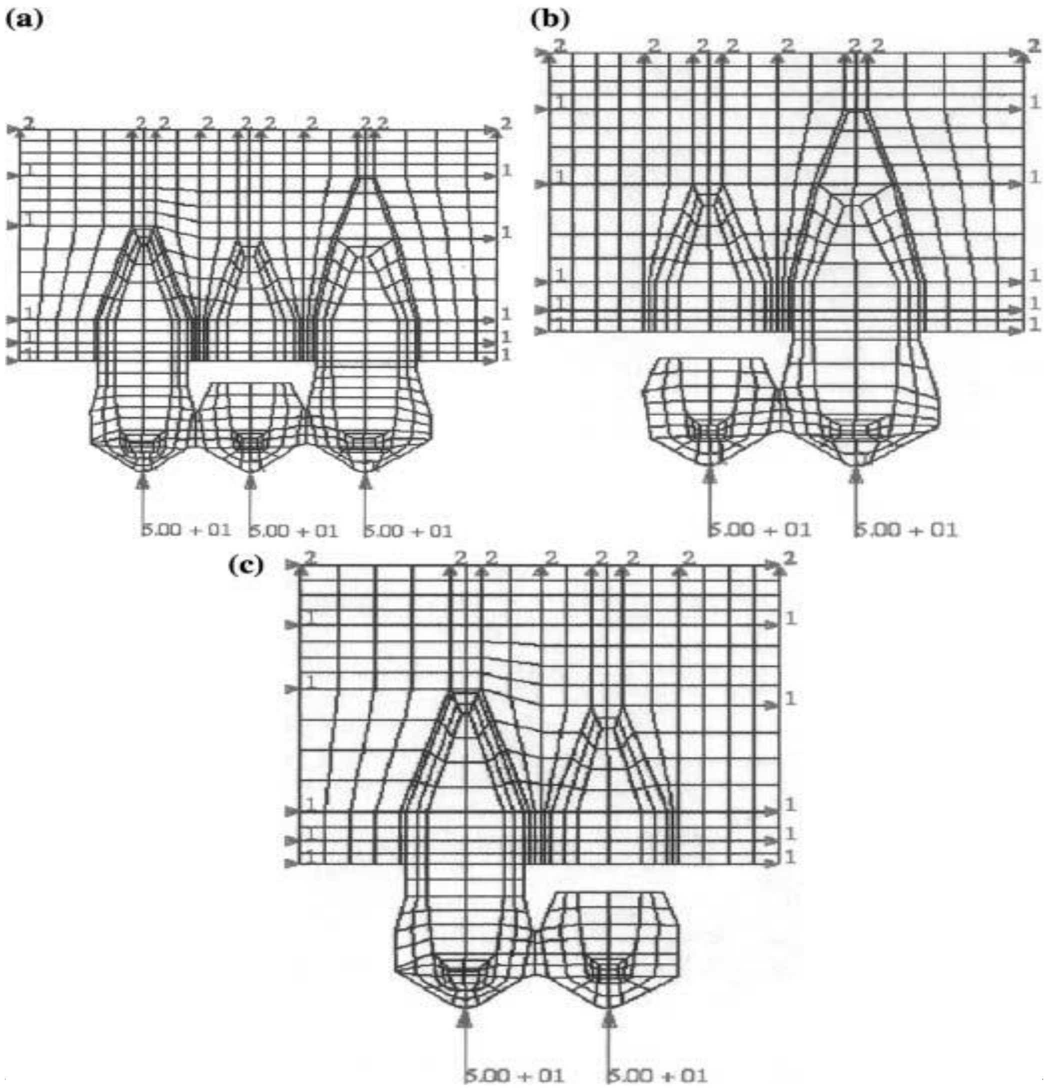

Fig 1: *Modelling 2D FEM*

Journal of Chemical Health Risks

www.jchr.org

JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

The mechanical properties of the cast made of gold combination and supporting structures [14] are summed up in Table 2. All used materials for the cast are assumed to be isotropic, homogenous, and linear elastic so that the variables throughout the FEA can be controlled. **For** the stress analysis, axial occlusal loads are applied in three unique manners in each FPD, giving an aggregate of 13 loading conditions. With the two-unit FPDs, pivotal occlusal heaps of force are applied midway to the distal unit.

With the two-unit FPDs axial occlusal loads of 50 N were applied centrally to the distal unit only (load 1), just to the mesial unit (load 2) and then to both the retainer and pontic simultaneously (load 3). Similarly, for the three-unit FPD, the scenarios were only one of the three units being loaded at any one time (loads 1, 2, and 3); two units being loaded simultaneously in three possible scenarios (loads 4, 5, and 6); and finally, all three units being loaded simultaneously (load 7).

For each 13 loading conditions, FPD displacement and maximum principal stresses are calculated using the software Abaqus (Version 5.8-1) and processed using Patran to show the outcomes as displaced shapes and stress contour fringes. The locations and magnitudes of the maximum displacements and the greatest maximum principal stresses were identified. The maximum principal stresses have been taken as the most positive direct stress acting on a plane with no shear stresses. For the avoidance of restoration failure, the maximum principal stress must be less than the critical value of any material. To evaluate the stress levels in each single component under different loading conditions, a 3D FEA model is created using the solid modeling software (Solidworks office 2007, Solidworks Corporation).

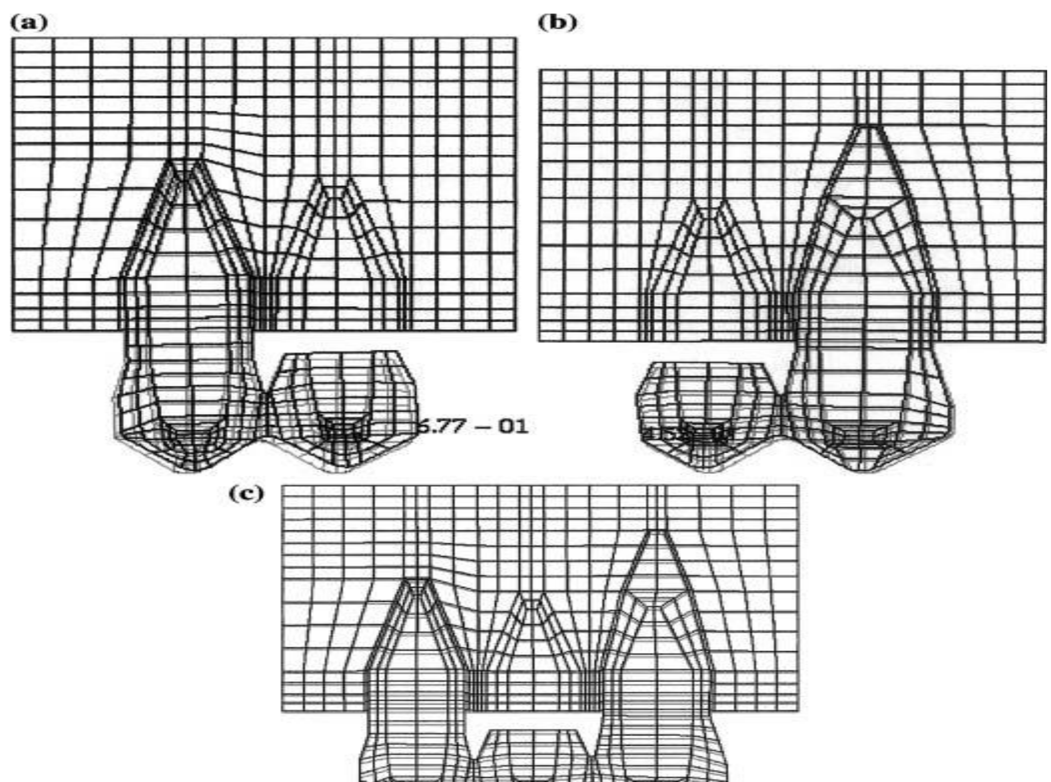

Fig. 2. *Removal of the fixed partial denture under occlusal stacking (a, mesial cantilever; b, distal cantilever; c, fixed partial denture replacement).*

Journal of Chemical Health Risks

www.jchr.org

JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

The framework was developed according to the manufacturer's instructions and guidelines on Y-TZP coping (values according to in-house testing, NobelProcera, Nobel Biocare AB, Goteborg, Sweden). The round edge framework with 0.6 mm thickness consists of an occlusal veneering thickness of a minimum of 1 mm to a maximum of 2 mm. The minimum connector area for a three-unit bridge supported by two implants placed in the posterior area, up to 20 mm away from each other, is 9.4 mm2 (height x width = 4.0 x 2.5). The design of the core involving approximal contact areas assures an appropriate support of marginal ridge allowing a veneering strength of 0.7 - 1.5 mm.

The implant geometry and abutment designs are obtained from 2D drawings of the implant manufacturer (BTLock srl, Vicenza, Italy). Two titanium fixtures 4.50 x 13 mm were selected for this study. A 5.90 mm long abutment with 4.70 diameter, 6 degrees of axial taper, and a 1 mm depth radial slight chamfer shoulder is used. This marginal design transmits less stress concentration (15).

The physiologic conditions could have been approximated by simulating the mandibular body, but the study aimed to evaluate only the biomechanical reliability of the prosthetic device. Therefore, bone volume was not considered and a smaller model was proposed for FEA using software ANSYS 7 (ANSYS Inc., Canonsburg, Pennsylvania, USA). The mesh was developed by SOLID187 elements, three-dimensional elements formed by 10 knots and very suitable for developing mesh on irregular bodies. It was achieved by a stress convergence analysis (through successive refinements with a σmax change to more than 10%) and it was divided in 224.620 elements connected at 375.629 points known as nodes (Fig. 4). The displacement of each of the nodes had to be calculated to determine the stress throughout the structure.

A finer mesh was generated at the material interface to ensure accuracy of force transfer. All materials used were assumed to be linearly elastic, homogeneous and isotropic. The mechanical properties were data supplied by the manufacturers and are showed in Table 2. An ideal osseointegration model is simulated in this research, so the fixtures are rigidly anchored along their entire interfaces. A fixed constraint on the external thread is applied, whereas a non-threaded portion is applied in the xz directions for the tangential movement.

Table 2. Mechanical Properties of Prosthetic Materials in Finite Element Analysis Evaluations.

Material	E [MPa]	G [MPa]	V	Fatigue [MPa] R=-1	Manifacturer
Y-TZP	210000	80769	0.33	550 n° of Cycles 1.00e+6	NobelProcera, NobelBiocare AB
Cement	5100	2008	0.27		RelyX ARC, 3M espe AG

Journal of Chemical Health Risks
www.jchr.org
JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

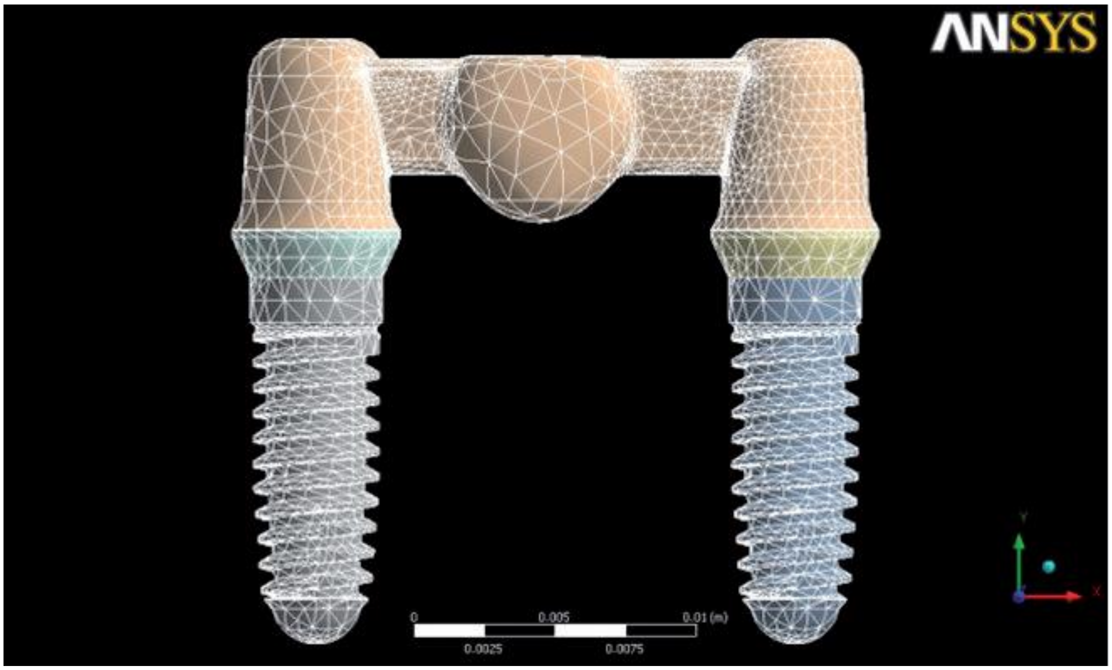

Figure 3 : *Three-dimensional finite element Y-TZP FPD model mesh.*

To observe the linear elastic behavior of materials, static-structural analysis is performed whereby three different angular directional (0°, 15°, and 35°) loads are applied. Axial and oblique loads of 100 N and 300 N were applied equally to the nodal points on the buccal side over a 0.5 mm^2 area on the stamp cusps. Four different loading conditions were applied: loading of all teeth to simulate maximum centric occlusion contacts, and loading of the single element of the framework to simulate single posterior contact.

Stress distribution within the elements was expressed in terms of von Mises equivalent stress and was compared with the yield strength values (29, 30). Calculated numeric data were transformed into color graphics for better visualization of the mechanical phenomena in the models. Determined that maximum von Mises equivalent stress, seen as the failure criterion, calculated on all elements, was lower than the yield strength value of the material, a further analysis was carried out. In particular, the maximum value was used as a reference value for the following fatigue analysis.

Result:

Fatigue analysis shows the linear elastic behavior of the dental implant before its breaking point. The observed von Mises stress values confirm that loaded areas bear the maximum stress in the marginal region of the prosthesis while occlusal regions of the prosthesis have the lower stresses. So, it can be stated that stresses originate in the occlusal region and increase in the gingival area. The highest stress values were located at connectors and cervical regions of abutments. Occlusal and gingival embrasures of connectors were the areas of more intensive stress concentrations with the higher stress observed at the cervical embrasure of the connector, between the pontic and abutment. Two different behaviors were observed in the stress distribution when the load was applied to the pontic element or to the pillars. The maximum stress was located at the connector regions of the bridge because it bended when the pontic element was loaded (Fig. 2). In the other two cases the maximum stress was found on the abutment of the pillar loaded (Figs. 3,4). Tensions rose with increasing inclination of the load (Figs. 5, 6). Y-TZP physical data were obtained from literature (31-34). Y-TZP

Journal of Chemical Health Risks

www.jchr.org

JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

shows very high static and cyclic fatigue strenght due to its polycrystalline structure.

Following values were obtained from tests carried out on Y-TZP samples (35-37):

σ_R 1000 MPa (fracture tension), $\sigma_{a\infty}$ 550 MPa considering several cycles of 2 x 10^{a6}

where $\sigma_{a\infty}$ is the fatigue strength limit and (R= -1) is the ratio cycles (2 x 106)

From these information accurate worth $\sigma_{a\infty,-1}$ can be discovered utilizing the accompanying equation:

$$\sigma^*_{a\infty,-1} = \frac{\sigma_{a\infty}(R=-1)}{K_f K_d K_l K_v} = \frac{550}{1 \cdot 1 \cdot 1.15 \cdot 1} = 478{,}3MPa$$

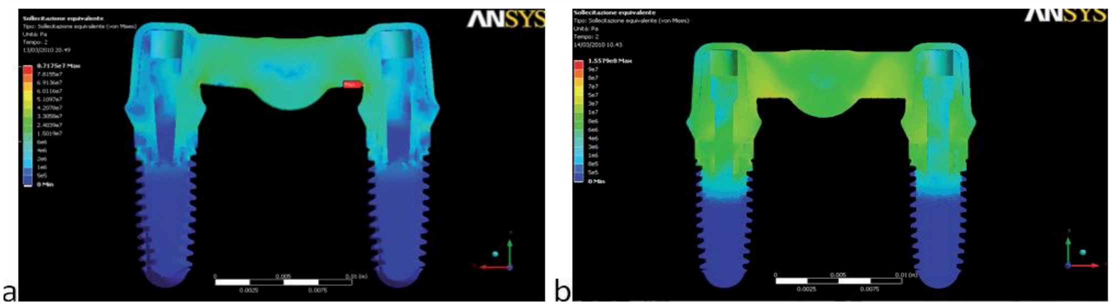

Figure 4 *von Mises pressure esteems (Pa) and dispersions found when the pontic load is applied: (a) 110 N – 0°; (b) 330 N – 45°.*

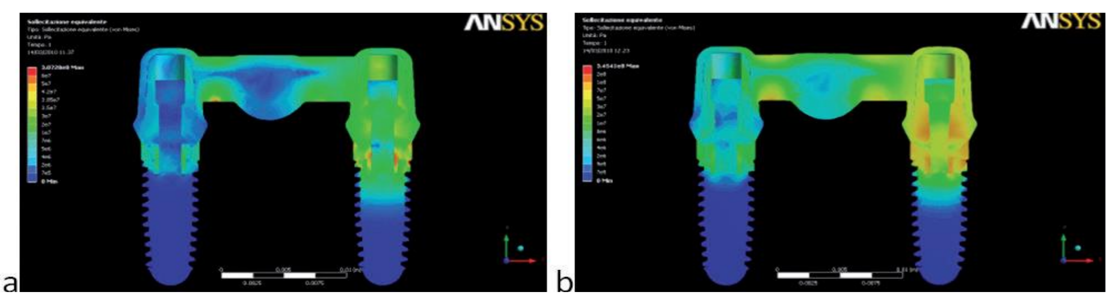

Figure 5 *von Mises pressure esteems (Pa) and dispersions found when the pontic load is applied (back stacking): (a) 330 N – 0°; (b) 330 N – 45°. High pressure ascends in both connector areas additionally when the heap is applied on one column pontic.*

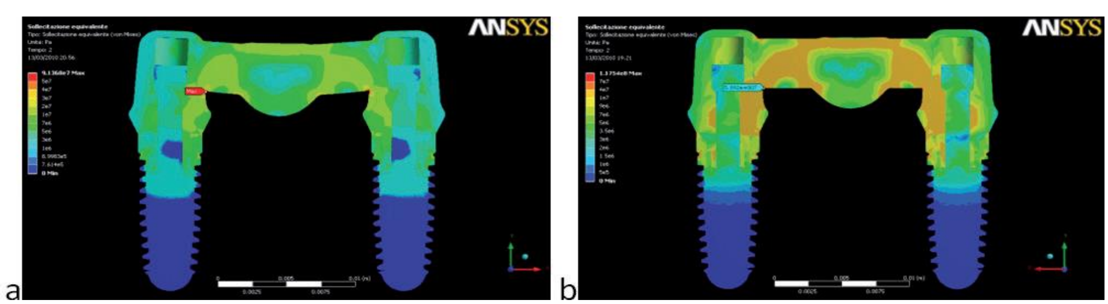

Figure 6 *von Mises pressure esteems (Pa) and circulations found in the physiological load: (a) 110 N – 0°; (b) 110 N – 20°. Connector part show greatest stress.*

Journal of Chemical Health Risks

www.jchr.org

JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

where K's are factors which impact Y-TZP weakness life (K_f shape, K_d size, K_l surface completing, K_v stress type).

Since, biting capacity is established by exchanging stacking cycles and diverse stacking designs, whose qualities fluctuate from least pressure (σ_{min} =0), to greatest pressure (σ_{max}), $\sigma_{a\infty}$ (R= – 1) got from test information, must be converterd into $\sigma_{a\infty}$ (R= 0) and the past equation into:

$$\sigma^*_{a\infty,0} = \frac{\sigma_R \cdot \sigma^*_{a\infty,-1}}{\sigma_R + \sigma^*_{a\infty,-1}} = \frac{1000 \cdot 478{,}3}{1000 + 478{,}3} = 323{,}5 MPa$$

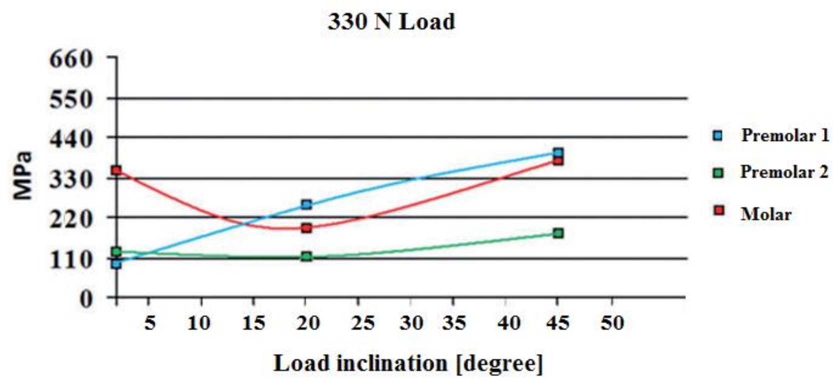

***Figure** 7 Dispersions of von Mises Stress (MPa) under 330 N load on single component of the system.*

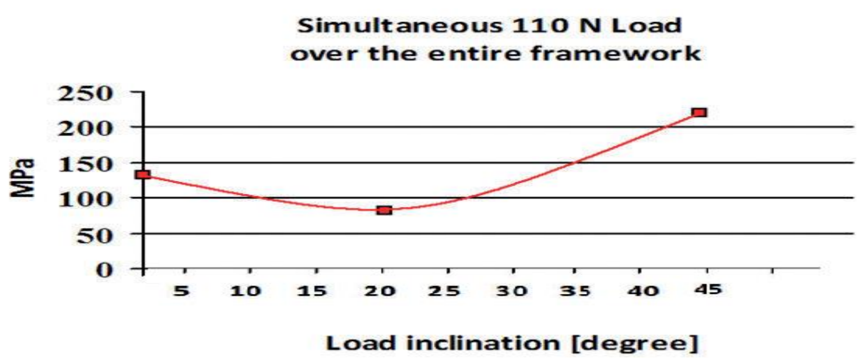

***Figure 8** Dispersions of von Mises Stress (MPa) under concurrent 110N load over the whole structure.*

Most extreme pressure esteems acquired from mathematical examination with various powers (110N and 330N) and points (0°, 20° and 45°) can be utilized for Y-TZP weariness investigation. As Table 3 appeared, no break weariness happened with a 110 N power, in light of the fact that $\sigma_\alpha < \sigma_{\alpha\infty,0}$. Accordingly, no system break happened after a boundless cycle number. Applying a 330 N power to the pontic no weariness issues happened in light of the fact that the heap is similarly upheld by entire framework. A 330 N power applied to one of the two columns, or to the two inserts produces weariness break (Table. 3).

Discussion:

The implant material has to be biologically safe and reliable and needs to have the best static and dynamic stability, which can be found in TZP ceramics making it a biocompatible ceramic material. Table 4 compares the different properties like Young's modulus, strength, and hardness of different materials. From Table 4, it can be observed that TZP provides

Journal of Chemical Health Risks
www.jchr.org
JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

the best and suitable properties of a ceramic which should be used as an implant material.

Table 3 *Maximum pressure and weakness quality qualities acquired in the reproductions. Red data demonstrate prosthesis greatest pressure (σ_{max}) and weakness quality cutoff ($\sigma_{a\infty,0}$) controlled by biting capacity rotating cyclic stacking.*

	Angle	Max Tension (F=110N)	Max Tension (F=330N)	*a*,max	**a∞,0**
PREMOLAR1	0	27.3	81.8	40.9	166.8
	20	70.8	212.5	106.25	166.8
	45	121.1	363.2	181.6	166.8
PREMOLAR2	0	32.9	98.6	51.4	323.5
	20	33	102.8	51.4	323.5
	45	51.8	155.7	77.85	323.5
MOLAR	0	102.4	307.3	152.65	166.8
	20	44.2	174.7	87.35	166.8
	45	115.1	345.4	172.7	166.8

Table 4: Comparison of properties of different materials used as biomedical materials

Property	Units	Ti 6Al 4V	316 SS	CoCr Alloy	TZP	Alumina
Young's modulus	GPa	110	200	230	210	380
Strength	MPa	800	650	700	900-1200	>500
Hardness	HV	100	190	300	1200	2200

Whether the restorative material will be a success or failure, will depend on the material property to withstand and resist occlusal forces. Examining and observing the behaviour of dental implants requires a complex simulation of the system [3] and many alternative experimental procedures like photoelasticity, load-to-failure bench-top testing, and the strain-gauge methods can be found in the literature but these are not precise and reliable, but FEA technique can give reliable stress analyses to predict the success rate of an implant. The FEA technique is very popular because it can assess the complex irregular prosthetic structures precisely. The FEM is a useful tool to investigate the oral systems during *in vitro* and *in vivo* investigations (3). The materials were all assumed to be homogenous, isotropic and to possess linear elasticity. Thus, the inherent limitations of this study should be considered. When applying FEA to prosthesis, it is important to take in consideration not only axial loads but horizontal (moment-causing loads) and combined forces (oblique occlusal force) too. This stress pattern is very close to physiological occlusal loading and will result in localized stress at the cervical area and the implant neck. The nodal points of load application were on the buccal side of the framework with an occlusal force of 100 N and 300 N.

Journal of Chemical Health Risks

www.jchr.org

JCHR (2023) 13(6), 2436-2446 | ISSN:2251-6727

The stress values are lower than the material strength of the applied materials. The results showed that maximum stress value trend was the same for the two loading conditions applied (100 N and 300 N), because constraints and geometry were the same for the two analyses. The structure response was the same, while maximum stress values varied. Tension values increased when the angles of oblique load became larger (35°), probably due to a flexural component that accentuated the stress peak on the implant.

An exception could be found when the load was applied to the pontic of the bridge.

The simultaneous 300 N loads applied on the molar (under the two stamp cusps) was the most critical situation obtained in the simulations. High stress always occurred throughout the marginal area of the prosthesis. Stress increased from the occlusal level toward the marginal area. Slight chamfer could be the best geometry to minimize the stress. Smooth and round framework design is indicated to increase the resistance to fracture.

Thus, a prosthesis, because continuously subjected to alternating loads due to chewing, has a greater long-term performance with more extensive connections. Prostheses supported by 1 or 2 implants replacing missing posterior teeth are subjected to an increased risk of bending overload (39). The type of loading may influence the stress patterns developed. Research has shown that the connector is the weakest region of an FPD (40-47) even if most of this research applied load just at the center of the pontic (40, 41). In the physiological pattern load is distributed to the entire surface of the restoration involving all the main parts: pillars and pontic. The weakness of connector area and its fracture strength to the fatigue stress depends on connector length, width and height. The functional and aesthetic customized design of the framework oblige to different sizes of the two connectors, with premolar's one shorter. Higher stress values resulted in the longer connector molar region. Therefore, it is mandatory to increase width and height in proportion with the increasing length.

Conclusion:

In this study, FEA technique was used to compare stress distribution on the Y-TZP frameworks of 3-unit implant-supported FPD. Different types of loading were applied to the framework. The validity of FEA results depends on the precision with which the geometry, material properties, interface condition, support, and loading are by physical reality.

In the current study, load was applied over the entire framework surface in correspondence of the two pillars and the pontic element. Results confirmed the vulnerability of both connector areas even if just one pillar was loaded. The high elastic modulus of YTZP, used in the present study, ensures a uniform distribution of biomechanical stress within the framework, providing more efficient load transfer and long-term predictability of the restoration. The primary cyclic fatigue evaluation indicates a strong propensity for fatigue behaviour, presenting a considerable range of loading conditions, where cyclic fatigue can be detected.

References:

1. Kelly JR, Tesk JA, Sorensen JA. Failure of all-ceramic fixed partial dentures in vitro and in vivo: analysis and modeling. J Dent Res 1995; 74:1253-8.
2. Crandall SH, Lardner T, Dahl NC. An Introduction to the Mechanics of Solids, 2nd Edition. New York, NY: McGraw-Hill, 1978.
3. Kamposiora P, Papavasiliou G, Bayne SC, Felton DA. Stress concentration in all-ceramic posterior fixed partial dentures. Quintessence Int 1996; 27:701-6.
4. Piconi C, Maccauro G. Zirconia as a ceramic biomaterial. Biomaterials. 1999; 20:1-25.
5. Rangert B, Krogh PH, Langer B, Van Roekel N. Bending overload and implant fracture: a retrospective clinical analysis. Int J Oral Maxillofac Implants 1995; 10:326-34.
6. Hauptmann H, Suttor D, Frank S, Hoescheler H. Material properties of all-ceramic zirconia prostheses. J Dent Res 2000; 79:507.
7. Rountree P, Nothdurft F, Pospiech P. In-vitro investigations on the fracture strength of all-ceramic posterior bridges of ZrO2-ceramic [abstract]. J Dent Res 2001; 80:57.
8. Von Steyern PV, Jonsson O, Nilner K. Five-year evaluation of posterior all-ceramic three-unit (In-Ceram) FPDs. Int J Prosthodont 2001; 14:379-384.

9. Raigrodski AJ, Chiche GJ. The safety and efficacy of anterior ceramic fixed partial dentures: a review of the literature. J Prosthet Dent 2001; 86:520-5.
10. Filser F, Kocher P, Weibel F, Luthy H, Scharer P, Gauckler LJ. Reliability and strength of all-ceramic dental restorations fabricated by Direct Ceramic Machining (DCM). Int J Computerized Dent 2001; 4:83-106.
11. Oh W, Gotzen N, Anusavice KJ. Influence of connector design on fracture probability of ceramic fixed partial dentures. J Dent Res 2002; :623-7.
12. Oh W, Anusavice KJ. The effect of connector design on the fracture resistance of all-ceramic fixed partial dentures. J Prosthet Dent 2002; 87:536-42.
13. De Aza AH, Chevalier J, Fantozzi G, Schehl M, Torrecillas R. Crack growth resistance of alumina, zirconia and zirconia toughened alumina ceramics for joint prostheses. Biomaterials. 2002; 23:937-945.
14. Romeed SA, Fok SL, Wilson NHF. Finite element analysis of fixed partial denture replacement. J Oral Rehabil 2004; 31:1208-17.
15. Sahin S, Cehreli MC, Yalcin E. The influence of functional forces on the biomechanics of implant-supported prostheses - A review. J Dent 2002; 30:271-282.