EGYPTIAN
DENTAL JOURNAL

Vol. 62, 1593:1603, April, 2016

I.S.S.N 0070-9484

WWW.EDA-EGYPT.ORG

STRESS ANALYSIS STUDIES WITH DENTAL IMPLANTS, DIFFERENT SITUATIONS AND CONNECTIONS WITH NATURAL TEETH

Emad M T Agamy *

ABSTRACT

The dental implants represent a useful method in solving many problems in restoring edentulous or partially edentulous patients. Their integration to the jaw bone can be predicated to large extent; however maintaining such integration depends on many biomechanical factors that should be considered. The implant position in relation to occlusal plane; the amount and direction of occlusal loads and connection to natural abutments are among these factors. In order to predict the clinical behavior of a specific implant configuration for treating certain partially edentulous or completely edentulous situations; several methods can be utilized in vitro. Strain gages and finite element analysis (FEA) methods are widely used for these purposes. The current study tries to cast light on some questions; linear or staggered arrangement of implants in completely edentulous situations. Is it safe to connect an implant to natural abutment with different bone support in over- denture design? Is it necessary to splint in the previous situations. Finally; what are the limitations of the most widely used stress analysis methods?

KEY WORDS: Dental implants, stress analysis, removable partial, dentures, overdenture, finite element analysis, strain gages

INTRODUCTION

The dental implants represent a useful method in solving many problems in restoring edentulous or partially edentulous patients. Their integration to the jaw bone can be predicated to large extent; however marinating such integration depends on many biomechanical factors that should be considered.

Success of the implants depends on implant stability which consists of two parameters: primary implant stability and biological stability. Primary implant stability refers to the stability of a dental implant immediately after implantation. The value of primary implant stabilization decreases gradually with reconstruction of bone tissue around the implant in the first weeks after surgery, leading to secondary stability. Secondary stability character is quite different from the initial stabilization, because it results from the ongoing process of osseointegration. When the healing process is complete, the initial mechanical stability is fully

* Assoc Prof; Acting Head of Prosthetic Dentistry Department, Minia University

replaced by secondary biological stability. If primary stability was not high enough following implantation, the implant's mobility is high which prevents osseointegration (secondary stability) leading to risk of implant failure [1].

One of the factors that affect implant stability; hence success, is the stresses generated and delivered to the dental implant and its surrounding structures. Even if osseointegration is achieved a possibility of failure still exists if the generated stresses around the dental implant surpass the bone remodeling capabilities. This hypothesis was confirmed by animal studies, in which complete or partial loss of osseointegration was found around the implants axially loaded excessively [2, 3]. The involvement of over stresses is also supported by another animal study [4] in which cyclic axial tension within biting force (non-axial), led to bone resorption around the implant neck. In unloaded implants used as control, no bone loss was observed. In the aforementioned studies, bone strains beyond the acceptable threshold have been suggested to cause the bone loss

Due to this fact several studies and methods are found in the literature addressing bone implant stresses. These methods are: Brittle lacquer method that was used only for detecting surface stresses; Theoretical equations that were applied for simple geometry analysis and Mechanical dial gages that were used mainly to measure biting force.

Other more complex methods are stereophotogrammetry (which is now used for topographical analysis or even CAD CAM especially for facial reconstruction [5]), holographic interferometry (is more used now for laser scanning as part of the CAD CAM procedures), photoelasticity, finite element analysis, and electric resistance strain gauge.

Photoelastic stress analysis is a modeling method that uses the optical effect of double refraction of mechanically or thermally loaded transparent resin for analyzing stress [6]. Photoelastic analysis has a lot of advantages as: it furnishes full-field values of the principal stress directions (sometimes called stress trajectories), it is adaptable to both static and dynamic investigations, requires only a modest investment in equipment and materials for ordinary work, and fairly simple to use [7].

In spite of the several advantages obtained through the use of photoelastic technique, it has several disadvantages. The first is the difficulty in obtaining stress free experimental model which influences the validity of the collected data in terms of stress position and magnitude. Moreover the elasticity of the tooth structure cannot be duplicated, the tensile properties of the human periodontal membrane cannot be simulated and finally the physical properties of plastics are of uniform density while bone is not uniform [8].

Also it requires that a model of the actual part be made of special materials (unless photoelastic coatings are used). It requires rather tedious calculations in order to separate the values of principal stresses at a general interior point. It requires expensive equipment for precise analysis of large components. It is very tedious and time-consuming for three-dimensional work [9].

Two and three dimensional finite element stress analysis has been used extensively in dentistry. Finite element analysis (FEA) is based on formation of a model that is divided into finite number of similar elements of the same geometrical shape; these elements are given the actual properties of the real material including modulus of elasticity and Poisson's ratio. Each element is inter-connected to its neighboring elements at a number of discrete points called nodes. When these nodes are subjected to the anticipated loading conditions, they result in a behavior of the model similar to the structure it represents. The displacement of each of the nodes is calculated to determine the stress throughout the structure. The mechanical behavior of each element can be expressed as a function of displacement of the nodes [10].

FEA provides several advantages such as – readability to handle a relatively complex geometry, complex restraints (Indeterminate structures can be solved) and can perform complex loading: Nodal load (point loads), Element load (pressure, thermal, inertial forces), time or frequency dependent loading. However, FEA technique has some limitation and obtains only “approximate” solutions. The FEM has “inherent” errors and the Mistakes by users can be fatal [11].

Strain gauge method is the most common technique used in experimental mechanics to evaluate strain at a point. A strain gauge is a device used to measure the strain of an object. The most common type of strain gauge consists of an insulating flexible backing which supports a metallic foil pattern. The gauge is attached to the object by a suitable adhesive, such as cyanoacrylate. As the object is deformed, the foil is deformed, causing its electrical resistance to change. This resistance change, (usually measured using a Wheatstone bridge), is related to the strain by a quantity known as the gauge factor[9].

The strain sensitivity is evaluated as a function of relative change in dimension and in the basic resistance of a material when it is stretched under load. The idea of electric strain gauge analysis depends on application of a metal element, such as a wire, that will change its resistance when it is elongated[12].

Ideally the gauge should have a high gauge factor which means that a small strain gives a large change in the resistance. It should also have a high degree of thermal stability in order to minimize apparent strains due to transient thermal fluctuations [13]. Finally it is also important to protect the gauge from humidity to obtain a reliable reading [14].

The widely used types of electric strain gauges are the bonded wire and the metal foil strain gauges. The bonded wire strain gauges consist of a fine wire laid in zigzag fashion and sandwiched between two strips of paper. In the metal foil strain gauges, a very thin foil is used instead of the fine wire which has greater heat dissipation properties [15].

Latest studies published with strain-gauge analysis show the use of this method to examine the biomechanical aspects of dental implant with different attachment system, to measure the force transmission onto implants and to assess the deformation of abutments of different heights in mandibular implant-supported overdentures [15-18].

It is still a matter of controversy which method of stress analysis can be used for each study or simulated clinical situation. The aim of the current work is to explore the possibility and sensitivity of two of the most used stress analysis methods in dental implants field (FEA and strain gauges).

MATERIALS AND METHODS

Part 1: stress analysis by finite element in three implant supported mandibular overdenture: to test the effect of changing the implant arrangement on stress distribution in mandibular overdenture cases. A finite element model was drawn using Solidworks software (Solidworks Corporation, Massachusetts, USA) simulating an edentulous mandible. Three implants were drawn in a linear configuration (testing condition A) or in staggered configuration (testing condition B). The implants (13 mm length 3.7 mm diameter, Screw plant ™, Direct ™ implant, The Spectra-System, USA) were drawn in bone cylinders each 6mm diameter to simulate a complete osseointegration. i.e full contact with no slippage possibility. Each implant with its corresponding cylinder was embedded in the model as one complex. The outer surface of the mandible, symphysis area, middle bone cylinder and coronal 2mm of the remaining cylinders were assumed to be pure solid cortical bone. The remaining parts of the simulated mandibular model and bone cylinders were assumed to be homogenous cancellous bone. The denture and attachments were also simulated with the mucosal coverage of the ridge (fig1).

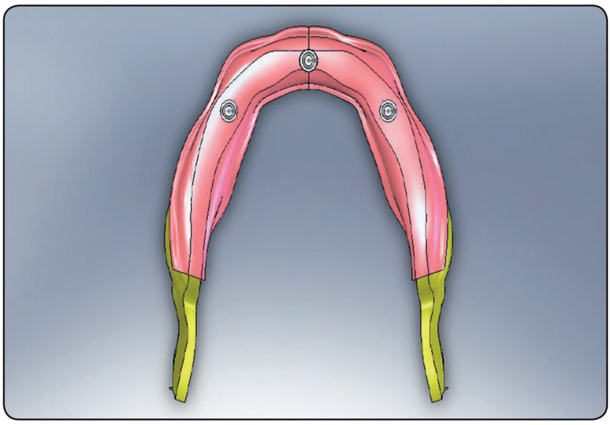

Fig. (1) The simulated model showing staggered arrangement of the implants

The material properties (table 1) and boundary conditions were defined as well as loading amount and direction (fig2 and 3). The testing conditions were defined and the tests were carried out using ANSYS software. The simulated load was 150 N distributed in axial or oblique fashion over the functional cusps of first, second premolars and first molar of the right side.

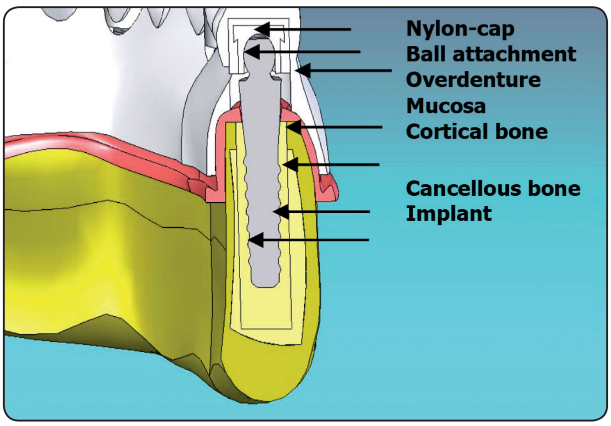

Fig. (2) The different parts of the simulated model and their nature of contact

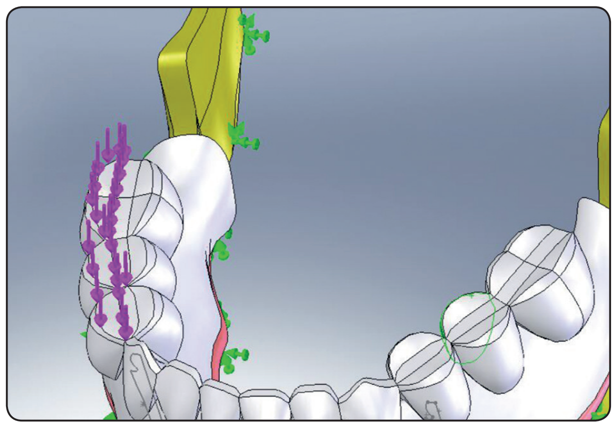

Fig. (3) Loading conditions and model restraints

TABLE (1) Modulus of elasticity and Poisson's ratio of the materials of the simulated model.

Material	Component	Material properties	
		Modulus of elasticity (GPa)	Poisson's ratio
Compact bone	Outer parts, symphysis, and coronal 2mm of bone cylinders	15	0.30
Cancellous bone	Remaining parts of the mandible	1.50	0.30
Titanium	Implant	110	0.33
Acrylic	Overdenture	2.7	0.35
Nylon	Nylon cap	0.17	0.40

Part 2: strain deformations around an implant attached to a natural abutment in implant supported mandibular overdenture: a physical model was constructed from heat cured acrylic resin similar to a technique described in the literature 19. This model simulated a partially edentulous mandible with remaining left canine. The periodontal ligament around the canine and the mucosal coverage of the mandible were simulated using poly vinyl siloxane material (Silaplast future and Silasoft–Normal, DETAX,GmbH ,Ettilgen, Germany). An implant was inserted in the right canine area 13 mm lengths, 3.7mm diameter (Screw plant ™, Direct ™ implant, The Spectra-System, USA) and strain gages (Kyowa strain gauge, KFG-3-120-c1-11l1M2R, Japan) were attached around the canine and implant (mesial and buccal to each) fig 4.

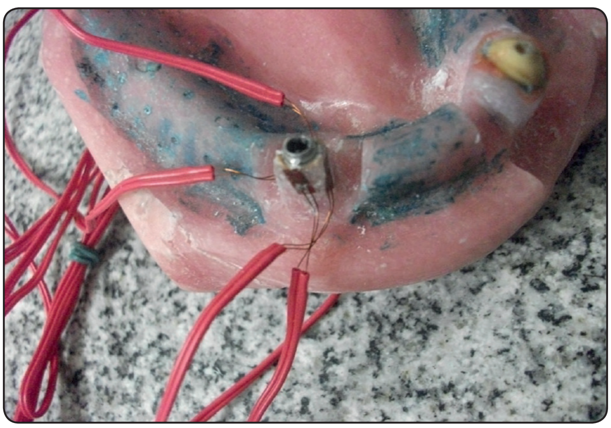

Fig. (4) The simulated model with strain gauges attached around the implant

The canine was reduced to simulate overdenture abutment preparation, and a channel was created in the root portion to accommodate a retaining screw to fix the bar later on through a metal coping. A castable abutment was tightened to the implant and a plastic bar pattern was attached to it and to the canine by burnout resin material (Duralay, Reliance Inc, Chicago, IL, USA). The bar construction was unscrewed from the implant and canine tooth and a duplicating mold was constructed from additional silicone material (Silasoft). A replica of the bar and copings was created in the duplicate mold by resin material (Duralay). The copings of this replica were separated from the bar and cast on with the original bar copings complex in the same ring. These procedures facilitated the interchangeability between the two designs during loading.

The loading conditions simulated were: at the actual bone support around the natural abutment, with 3, 6 and 8 mm bone loss around the abutment either splinted by a bar or fitted with solitary attachment figs 5and 6. The applied load was static pointed load of 60 N at the tooth side and implant side of the denture separately. The load was applied into a prepared point at the occlusal surface of first molar during splinting the canine with the implant as step one of testing and during separate attachment connection as step two. The direct loading on the canine tooth of the model showed the direction of compressive microstrains which were assigned the (+) sign and the tensile microstrains were assigned the (-) sign.

The collected data were arrayed and statistically tested (SPSS version 16, SPSS Inc, Chicago. IL, USA) using one way ANOVA for the 4 strain gauges position and student t- test for attachment condition (splinted or not).

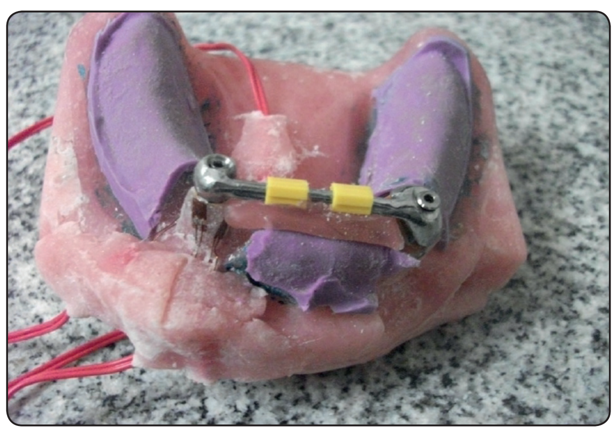

Fig. (5) Bar attachment and zero bone loss around the canine.

Fig. (6) Solitary attachment and 8mm bone loss around the canine

RESULTS

The results of part one are shown in fig. 7 and 8. The pattern of stress distribution clearly indicated that;

For vertical load:

Tripodal implants arrangement gave higher stress values at all implant sites (Loaded, median & non loaded implants) than linear arrangement. Tripodal implants arrangement gave higher stress values at overdenture than linear arrangement and it showed the highest stress value in the model. Linear arrangement gave higher stress value at distal bone at loaded site than tripodal arrangement.

For oblique load:

Tripodal implants arrangement gave higher stress values at Loaded and median implant jaw bone sites than linear arrangement. Linear arrangement gave higher stress value at non loaded implant and overdenture than tripodal arrangement. The highest stress value recorded at loaded implant at attachment-implant connection (i.e implant platform).

The results of part two of the study showed that (figs 9-10): with full bone support or only 3mm bone loss; splinting leads to more share of load to the implant and more homogenous distribution. With 6mm bone loss around the natural teeth; splinting favors distribution around the natural abutment but an extrusion component is developing around the implant. With 8mm bone loss; un-splinted configuration favors better stress distribution. The detailed results showed that the recorded microstrains at the four strain gauges are significantly different than each other statistically (ANOVA).

The recorded microstrains around the tooth were in general tensile in nature while around the implant were mostly compressive in nature. Splinting has a significant effect on the natural abutment for different degrees of bone loss as it either decreases the values of recorded microstrains or changes their direction (student t test).

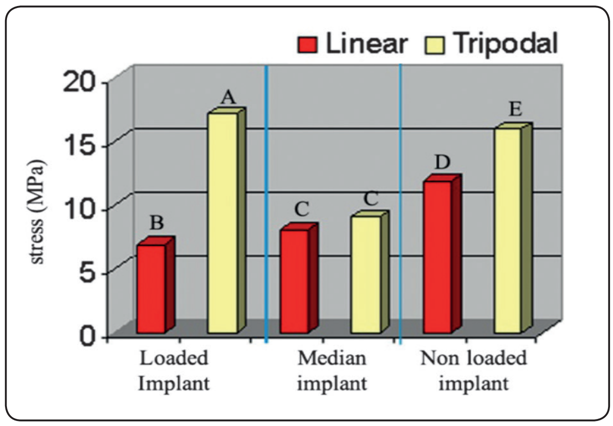

Fig. (7) maximum recorded stresses around the implants in linear or tripodal (staggered) situation for mandibular over dentures

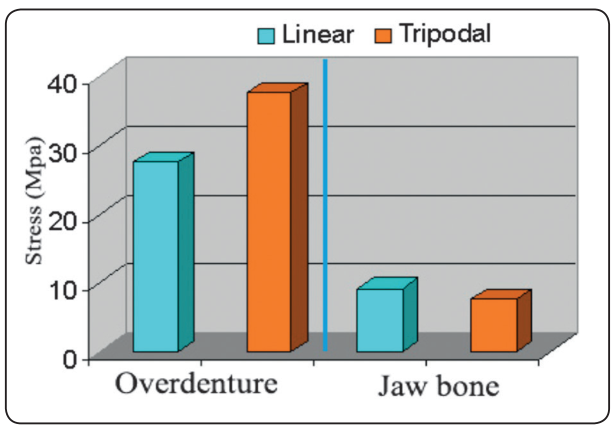

Fig. (8) maximum recorded stresses around the denture and jaw bone in linear or tripodal implant arrangement

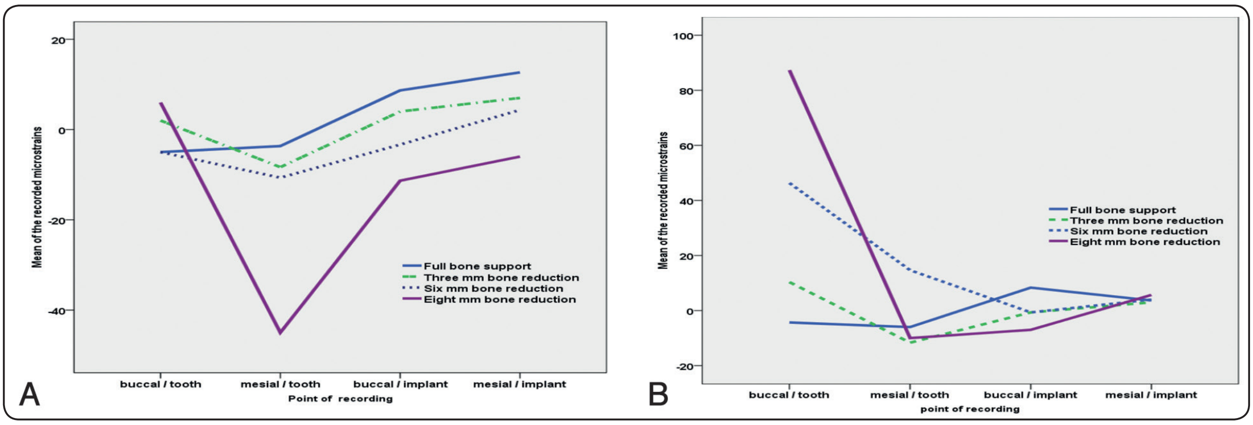

Fig. (9) Recorded microstrains around the supporting structures in case of loading at the canine side. A: splinted configuration and B: non splinted

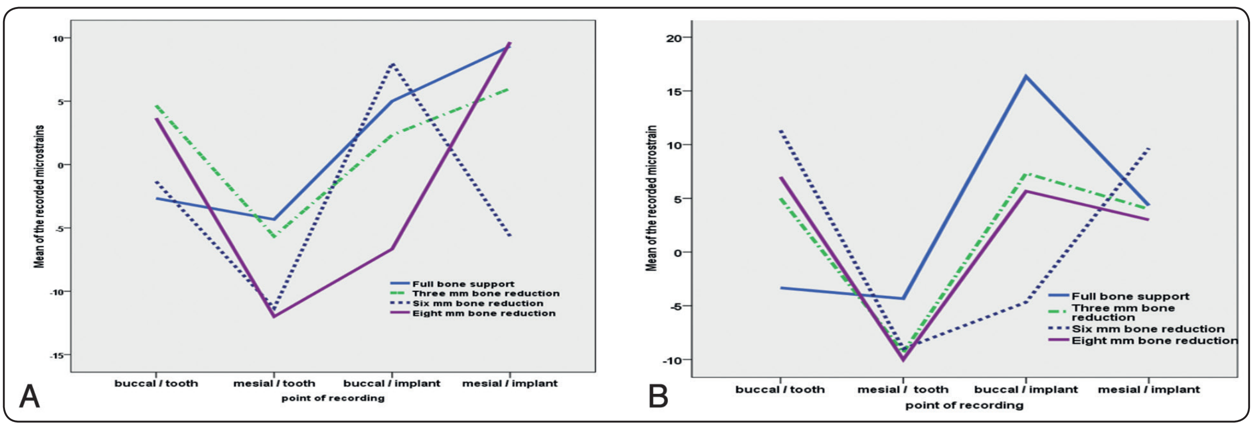

Fig. (10) Recorded microstrains around the supporting structures in case of loading at the implant side. A: splinted configuration and B: non splinted

While for the implant splinting produced slight changes in amount and pattern of recorded microstrains that were statistically insignificant for total bone support 3 or 6 mm bone loss. While for 8 mm bone loss splinting produced tensile microstrains opposite to the two bonded strain gauges around the implant which can be considered as extrusion forces generated around the implant.

DISCUSSION

The importance of stress analysis studies in the field of oral implantology cannot be neglected. These studies can predict whether a specific design (implant or prosthesis), a certain arrangement, or loading manner can be successful or not through measuring the stresses generated. These stresses can be greater than the bone remodeling capabilities especially when the dental implant is connected to a natural abutment tooth in partially or subtotal edentulous situation.

Clinical conditions vary considerably from one patient to another which makes applying randomized controlled clinical study the golden standard for valid clinical conclusion.

However in view of the limitations of conducting well controlled randomized clinical studies; one should rely on sound biomechanical rules during designing implant supported prostheses.

The current work addresses two of the most used technique of stress analysis studies; strain gages (SG) and finite element methods (FEA). Both techniques are accurate and reliable and were actually used alternatively to verify each other.

Variables in implant design or dimensions (length and diameter) as well as prosthetic design and material were found to affect stress distribution around dental implants which may lead to failure [20-24]. These findings were also confirmed by clinical reports and studies.

Finite element analysis (FEA) has been used to predict the biomechanical performance of various dental implant designs as well as the effect of clinical factors on their success.

The development of FE models requires specific knowledge of the mechanical properties of both the jaw bones and implants. Numerous studies have been conducted on the mechanical behavior of titanium implants, but determination of the biomechanical properties of living tissues or bone remains challenging. The wide range of values published in the literature for Young's modulus of human bones confirms this situation. More recent work with FEA modelling depends on simulation of the required model through acquisition of CT scan data of a volunteer patient [25, 26]. The obtained model can be theoretically sent directly FEA software but till now there must be third party software as a transitional step and probably manual drawing of some components due to software conflict. This new modification of FEA model construction has the potential to optimize the results obtained by FEA method to the level of individual planning for implant supported restorations once the conflict between data acquisition and 3D modelling software is resolved.

The only limitation that will remain then is the inability to simulate the mechanical behavior of the living human bone tissue and its response to applied mechanical forces [20]. If this can be effectively achieved it will lead to an increase in knowledge of stress distributions and magnitudes within the implant and surrounding jawbone that will lead to optimization of implant designs and insertion techniques.

Strain gauges (SG) provide several advantages in comparison to the other techniques of stress analysis specially FEA and photoelastic techniques. This method is simple and flexible, low cost and it is available in markets. Strain gauges measure the total strain occurring at the area they are attached to where one can calculate the stresses indirectly. Based on their dimensions and number; they can give a general idea or a detailed idea about the created stresses (magnitude and direction) inside a specific design. Strain gauges small enough to be attached to the implant neck are present in the market; however they need to be installed by the manufacturer, then one can build the physical model around them. In such case they may be destroyed or detached, also a complex equipment and data acquisition software will be needed. They may be used in-vivo or in-vitro. When they are used in vivo they are subject to humidity, breath and temperature changes that may render them inefficient. They are cemented to the outer surface of the structures they are attached to or in the form of transducers. So they cannot predict the internal strains (stresses).

Although SG method is the most commonly used for stress analysis, it has some limitations in the form of sensitivity to electric noise, high temperature, thermal and electromagnetic induced voltage interference which may alter the analysis readings [21]. Also it gives information that are valid only for the specific points of static loading and their respective areas of strain gauge location, therefore it is difficult to judge that the selected points have the best location for evaluation of stresses. It is also difficult to determine the magnitude and direction of the applied static load in order to mimic the chewing cycle pattern [22]. Moreover, the strain gauge technique allows only for point stress analysis, therefore a large number of gauges and sophisticated mathematical calculations must be applied to assess the distribution of stress [23].

The selection of FEA method for part one of the studies was very suitable as all the variables of the study were the same except the implant position (tripodal or linear configuration). The implant dimensions, attachment type, relation to superstructure and prosthesis material and design were all the same. The model dimensions, anatomical structures (and hence material properties) and relations as well as fixation points and constraints were also the same. If such study would be conducted using SG method it would be impossible to be conducted on the same model for many reasons. If the position of the strain gauges were fixed at the two posterior implants so the data regrading the anterior implant would be lost as in this case it would be the implant without a gauge. This implant would not be studied by strain gauge as it will be changed in position in the second part of the study. So if a gauge was bonded to it at the first position it must be changed and another one should be bonded to it at the new position. The readings obtained by the strain gauges are relative to their position [22]. Similar problem will exists if middle implant position was fixed and the change was obtained through the two posterior implants as the posterior strain gauges would have to be changed after testing the linear configuration. Another solution was to test the two configurations on two separate physical models using strain gauges, which could not be compared at all. It is very obvious that one cannot achieve two identical models in the macro and micro structures as well as their deflection pattern. It is also not possible in this case to adjust the simulated bone thickness around the implant after preparation to bond the strain gauges. Moreover unless ten or more models of each situation were constructed (to obtain standard error of the model construction technique) the results of each model will be valid for this model only.

The selection of SG for part two of the study was also very suitable for the studied situation. In this part reduction of the simulated bone support around the natural abutment was carried out in sequential manner. The gauges were bonded at two surfaces around the simulated canine at the lower half of the root. The reduction of bone was mainly from the other two surfaces so that the simulated abutment would remain in place. If this part was conducted by FEA it would have been very difficult to modify the model after finalization of the analysis at full bone support and new drawings for every situation would have been necessary. This in turn would have created the problem of several models rather than a modified single model. The only limitation of SG in this case is that the results are valid only for their relative position and full condition of stress distribution is lacking. However this limitation can be neglected as long as comparative study of two or more designs is considered on the same model.

Regarding the results of the first part of the study: the linear arrangement produces less total stresses where the implants share less in the support than the tripodal arrangement. The fact that the load applied, and the model design (including the implants and prosthesis) in both situations are the same implies that differences in reaction to the load will be only due to the different implant arrangement.

The linear arrangement constitutes a single beam that will not lock the movement of the three attachments mounted to the implants as they will have a common axis of rotation in the antero-posterior direction. The supra-structure (overdenture) will act as a cantilever that is free to rotate around the common axis of the implant attachments. This movement will be proportionate with the compressibility of the mucosa and is responsible for the greater percentage of stresses transmitted to the bone and the denture in this case. This assumed free movement of the denture explains also why the total values of stresses with linear arrangement are less than the tripodal arrangement. Applying the basic laws of mechanics can easily justifies that this movement constitutes a kind of mechanical work that will consume some of the load.

The nylon caps simulated for the attachments allowed for this assumed free movement of the denture as this material has the closest modulus of elasticity to the mucosa covering the residual ridge.

The tripodal arrangement on the other hand seems to lock the denture movement as there is an imaginary multi axial beam between the three implant attachments. The movement of this case will be resisted by the median and non loaded implant attachments and their nylon caps as reflected by the greater stresses delivered to them. The denture will act as a huge cantilever beam that is relatively locked. The consequences are increased amount of total stresses to the implants and their supra-structures as compared to the linear arrangement. In other words there is no mechanical work done that dissipates part of the load. The grave result is severe overloading of the loaded side implant in case of oblique load application that may lead to component failure over long period of function.

The oblique loading produced greater stresses than the vertical load in all situations. This is probably due to creation of multi axial movement of the denture that will be partly resisted by the median and non loaded implants. Limitation of the denture movement will create some friction at the attachment nylon cap level that will increase the stresses as a result of limited force dissipation. The results of oblique loading are especially important as most of the load applied to any prosthetic appliance will be oblique in nature as a result of the complex chewing cycle and occlusal anatomy.

When deciding to select a specific implant arrangement or prosthetic design one should take the possible damaging effects of oblique loading and their anticipated results. The results of the current study indicate that linear arrangement may be safer than tripodal arrangement for three implant placement in the anterior mandible. Such statements must be interpreted with caution as they only represent the modeled situation.

The results of the second part of the study on the other hand are mainly due to effect of splinting when comparing it on horizontal basis. i.e level of bone support. While the loss of bone support around the tooth will be the cause when viewing the results in longitudinal fashion. i.e when comparing microstrains pattern around the supporting structures when full bone support, 3, 6 or 8 mm bone loss.

Decreasing the bone support around the abutment tooth increases the possibility for its rotation under applied forces which changes the pattern of denture movement and may surpass the supporting capabilities of other structures. This is evident in the current study with bone loss 6 or 8 mm. Splinting on the other hand limits the movement of the natural abutment and creates a common linear anterior fulcrum line for the prosthesis to rotate around. This may limit the movement of the denture base and improve the microstrains pattern and amount. This seems to be valid only for the mild to moderate bone loss. For severe bone loss that results show possibility of extrusion components around the implants that will contra indicate maintaining the natural abutment and necessitate its removal.

CONCLUSIONS:

1. For a proper stress analysis using FEA, proper model construction as well as through knowledge of the material used is necessary.
2. For strain gauge method simulation of a model that represents the various mechanical properties of the studied structures (bone) is essential.
3. For three implant supported mandibular over denture linear arrangement may be better tolerated than staggered
4. For mixed tooth implant supported mandibular overdenture, splinting a natural canine with an implant may be beneficial as long as there is no sever bone loss around the tooth.

REFERENCES

1. "Implant stability", http://en.wikipedia.org/wiki/Primary_ implant _stability.
2. Isidor, F. Loss of osseointegration caused by occlusal load of oral implants. A clinical and radiographic study in monkeys. Clin Oral Impl Res. 1996, 7: 143–152.
3. Isidor, F. Histological evaluation of periimplant bone at implants subjected to occlusal overload or plaque accumulation. Clin Oral Impl Res. 1997, 8: 1–9.
4. Hoshaw SJ, Brunski JB, Cochran GVB. Mechanical loading of Brånemark implants affects interfacial bone modeling and remodeling. Inter J of Oral & Maxillofac Impl. 1994; 9: 345–360.
5. Heike CL, Upson K, Stuhaug E, Weinberg SM. 3D digital stereophotogrammetry: a practical guide to facial image acquisition. Head & Face Medicine 2010, 6:18. An open access article, BioMed central. http://www.head-face-med.com/content/6/1/18
6. Eisenmann E, Walter M.: Improving the fit of implant supported superstructures using the spark erosion technique. Int J Oral Maxillofac Implant 2004;19: 810-818.
7. Tada H, Paris PC, Irwin GR.. Stress Analysis of Cracks Handbook, 2nd ed1985. St. Louis, Mo.: Paris Productions, Inc.
8. McDowell G.C.: Force transmission by indirect retainers during unilateral loading. J. Prosthet. Dent. 1978; 39: 616-621.
9. Dally, J. W., and W. F. Riley. Experimental Stress Analysis, 3rd ed. New York: McGraw- Hill. 1991.
10. Rao SS "the finite element method in engineering http://www.slideshare.net/GourabChakraborty1/the-finite-element-method-in-engineering-er-gourabs-notes-on-engineering.
11. Mahesh "chapter 1:introduction to finite element analysis". http://www.scribd.com/doc/228042937/CAMA-Lab-Mechanical
12. Asundi A. and Kishen A: A strain gauge and photoelastic analysis of in-vitro strain and in-vitro stress distribution in human dental supporting structures. Arch Oral Bio; 2000, 45: 543-550.
13. Dally J.W. and Riley W.F.: Experimental stress analysis, McGraw Hill Book Co., London, 2nd ed. PP.88-112; 1978.
14. Maxfield J.B., Nicholls I.I. and Smith D.E.: The measurement of forces transmitted to abutment teeth of removable partial denture. J. Prosthet Dent; 1979, 41: 134-142.
15. Porter JA Jr, Petropoulos VC, Brunski JB. Comparison of load distribution for implant overdenture attachments Int J Oral Maxillofac Implants, 2002, 17 (5): 651–662
16. Maeda Y, Horisaka M, Yagi K. Biomechanical rationale for a single implant-retained mandibular overdenture: an in vitro study Clin Oral Implants Res, 19 (3) (2008), pp. 271–275.
17. Jacques LB, Moura MS, Suedam V, Souza EA, Rubo JH. Effect of cantilever length and framework alloy on the stress distribution of mandibular-cantilevered implant-supported prostheses Clin Oral Implants Res 2009, 20 (7): 737–741.
18. Suedam V, Souza EA, Moura MS, Jacques LB, Rubo JH. Effect of abutment's height and framework alloy on the load distribution of mandibular cantilevered implant-supported prosthesis Clin Oral Implants Res, 20 (2) (2009), pp. 196–200
19. El Charkawi HG and El-Wakad MT: Effect of splinting on load distribution of extra-coronal attachment with distal extension prosthesis in vitro. J Prosthet. Dent. 1996; 63(3):315 – 320.
20. Papavasiliou G1, Kamposiora P. Three-dimensional finite element analysis of stress-distribution around single tooth implants as a function of bony support prosthesis type and loading during function. J Prosthet Dent; 1996, 76(6): 633-640.
21. VAN STADEN R C, GUAN H, LOO Y C, Application of finite element method in dental implant research, Australia. Comput Methods Biomech Biomed Engin. 2006; 9(4):257-70.
22. Haldun I., Kivanc A. Murat C. Comparison of non-linear finite element stress analysis with in vitro strain gauge measurements on a morse taper implant. Int. J. Oral Maxillofac. Implants. 18: 258-265; 2003.
23. Elcharkawi H, Zekry K, Elwakad M. Stress analysis of different osseointegrated implants supporting distal extension prosthesis. J Prosthet Dent 1994; 72: 614-622.
24. Fernandes C, Glantz P, Svensson S. Reflection photoelasticity: A new method for studies of clinical mechanics in prosthetic dentistry. Dent. Mater. 19: 106-117; 2003.
25. Geramy A., Adibrad M., Sahabi M.: The effects of splinting periodontally compromised removable partial denture abutments on bone stresses: a three-dimensional finite element study. J. Dent. Sci. 2010 ; 5(1) : 1–7.
26. Agamy EMTM, Mohamed G F. Stresses around implants and natural abutments in free end RPD. EDJ April 2016 in press