International Journal of Science and Healthcare Research
Vol.5; Issue: 3; July-Sept. 2020
Website: ijshr.com
ISSN: 2455-7587

Original Research Article

Use of Finite Element Analysis for Predicting Fatigue Life of Implant and Its Correlation with Applied Forces

Desai Shivani Sharad[1], Harshad Nivrutti Kumbhar[2], Neeli Apoorva[3], Bhandary Shibani[4]

[1,2,3,4]Shiv Cancer Institute, Private Practice, Miraj, India.

Corresponding Author: Desai Shivani Sharad

ABSTRACT

Aim: To predict the fatigue life of an implant with the correlation of the applied forces by finite element analysis.

Methods and Material: The authors of the study have created a 3D model of the bridge supported by an implant. The implant material used was titanium alloy. A finite element analysis was conducted to calculate the maximum amount of stress and deformation formed in the implant and the bone on the application of a force of four different magnitudes- 250N, 500N, 750N, 1000N. By conducting a fatigue analysis, the minimum, average, and maximum amount of days the implant can survive under the applied forces was calculated.

Results: The maximum amount of stresses and deformation was observed at the abutment screw bone junction. The implant could withstand a force of 250N for an average of 8 years and a force of 1000N for an average of 2 years. With constant increase in the force, the lifespan of the implant decreased as the bone could not withstand the force.

Conclusions: Finite Element Analysis is a very useful tool to predict the mechanical properties of implants in dentistry and analyze the survival of the implant by calculating the fatigue analysis.

Keywords: Implants, Finite Element Analysis, Fatigue analysis

INTRODUCTION

Dental implants are a revolutionary contribution to the field of dentistry. The integration of the implant in the bone influenced by the quality and the quantity of bone and the implant loading have a major impact on the success rate of the implant. Forces of mastication can cause a certain amount of stress and deformation on the cortical and cancellous bone. [4, 9] Using finite element analysis we can calculate the amount of stress and deformation caused on the bone when a certain amount of force has been applied. The finite element method (FEM) is an approximation method, which replaces a complex structure with simple elements interconnected at points called nodes which can be assembled to represent any shape or defined model. [4,10] Each element can be assigned with material properties that are determined by the clinical situation or model conditions, and forces are applied to simulate clinical loads. [10] The experimental response to the applied forces or applied stress can then be visualized and calculated. [10] FEM allows for detailed visualization of strength and stiffness where structures bend and twist while indicating the distribution of displacements and stresses. [6, 8, 10]

MATERIALS & METHODS

In the study, the authors have performed the finite element analysis on the implant-supported bridge to study the stresses and deformation on the cortical and cancellous bone. A dummy crown material resembling the properties of the teeth was used over the implants. Titanium implant material was used for the following study. The force generated during routine mastication of food such as carrots or meat is about 70 to 150 N (16 to 34 lbs). [13] The maximum masticatory force in some people may reach up to 500 to 700 N (110 to 160 lbs). [12] Using FEM, the authors applied forces of 250 N, 500 N, 750 N, and 1000 N. The fatigue life of the implant was calculated in ANSYS.

Developing 3D model of implants in CAD software: Software used for Computer Aided Design (CAD) uses technology that designs a product and documents the design process. They produce virtual designs of the physical components and give us an Idea of final models. Software currently in use are Auto-CAD, CATIA, and Solid works, etc. In this research work, CATIA was used for developing a 3D model of an implant. At first two dimensional sketches of cortical and cancellous bones, bridge and implant were developed. Then they were converted into 3D models by using Rib features in software. All the three-dimensional components were assembled together to form the final structure ready for Finite element analysis. While exporting the file, it was saved in the format which is compatible with FEA software. [3] The figure below shows the 3D model of the prepared assembly.

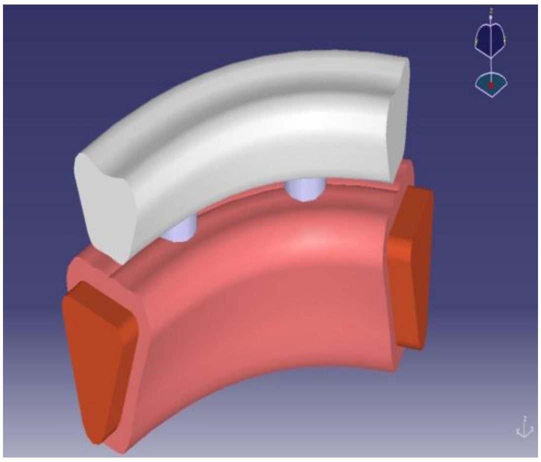

FIGURE 1: 3D MODEL OF IMPLANT WITH BRIDGE AND BONES ASSEMBLY

Selection and specifying the material and its properties: One of the initial steps in FEA software is to define the material properties as per the requirement. As the research involves cortical bone, cancellous bone, implant, and bridge, there is a requirement to specify respective material properties. Accordingly, all the mechanical properties of the material under the case study were enlisted, such as density, yield strength, hardness, Young's modulus, and Poisson's ratio, etc.

The below figures show the material module selected in the FEA software (ANSYS 16.0):

Properties of Outline Row 3: cancellous bone

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poisson...			
5	Young's Modulus	1.48E+10	Pa		
6	Poisson's Ratio	0.3			
7	Bulk Modulus	1.2333E+10	Pa		
8	Shear Modulus	5.6923E+09	Pa		
9	S-N Curve	Tabular			
10	Interpolation	Linear			
11	Tensile Yield Strength	1.29E+08	Pa		
12	Compressive Yield Strength	1.15E+08	Pa		
13	Tensile Ultimate Strength	1.4E+08	Pa		
14	Compressive Ultimate Strength	1.93E+08	Pa		

FIGURE 2: PROPERTIES OF CANCELLOUS BONE

Properties of Outline Row 4: Cortical Bone

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poisson...			
5	Young's Modulus	1.37E+10	Pa		
6	Poisson's Ratio	0.3			
7	Bulk Modulus	1.1417E+10	Pa		
8	Shear Modulus	5.2692E+09	Pa		
9	S-N Curve	Tabular			
10	Interpolation	Linear			
11	Tensile Yield Strength	1.29E+08	Pa		
12	Compressive Yield Strength	1.15E+08	Pa		
13	Tensile Ultimate Strength	1.4E+08	Pa		
14	Compressive Ultimate Strength	1.93E+08	Pa		

FIGURE 3: PROPERTIES OF CORTICAL BONE

Properties of Outline Row 6: Human teeth

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poisson...			
5	Young's Modulus	8.75E+10	Pa		
6	Poisson's Ratio	0.31			
7	Bulk Modulus	7.6754E+10	Pa		
8	Shear Modulus	3.3397E+10	Pa		

FIGURE 4: PROPERTIES OF HUMAN TEETH

Properties of Outline Row 5: Custom titanium

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poisson...			
5	Young's Modulus	1.1E+11	Pa		
6	Poisson's Ratio	0.35			
7	Bulk Modulus	1.2222E+11	Pa		
8	Shear Modulus	4.0741E+10	Pa		
9	Strain-Life Parameters				
10	Display Curve Type	Strain-Life			
11	Strength Coefficient	9.2E+08	Pa		
12	Strength Exponent	-0.106			
13	Ductility Coefficient	0.213			
14	Ductility Exponent	-0.47			
15	Cyclic Strength Coefficient	1E+09	Pa		
16	Cyclic Strain Hardening Exponent	0.2			
17	S-N Curve	Tabular			
18	Interpolation	Linear			
19	Scale	1			
20	Offset	0	MPa		
21	Tensile Ultimate Strength	434	MPa		
22	Compressive Ultimate Strength	848	MPa		

FIGURE 5: PROPERTIES OF IMPLANT

Meshing of 3D Model: For FEA analysis the whole assembly of implant and bone structure gets divided into a number of well-distanced points called nodes. These nodes are further joined to each other to form finite elements. The process of joining these nodes is called meshing. There are various types of mesh such as triangular, quadrilateral, pyramid, triangular prism, hexahedron, and polyhedron. For this research work hex dominant method was selected for meshing and mesh type was hexahedron. It was selected because it gives a fine quality of mesh and structure sustains its actual mechanical stiffness. While meshing elements size was chosen such that the difference between two nodes of an element was minimum. This selection also provides accurate results and to improve further, meshing was done without de-featuring with capturing all the curvatures and proximity.

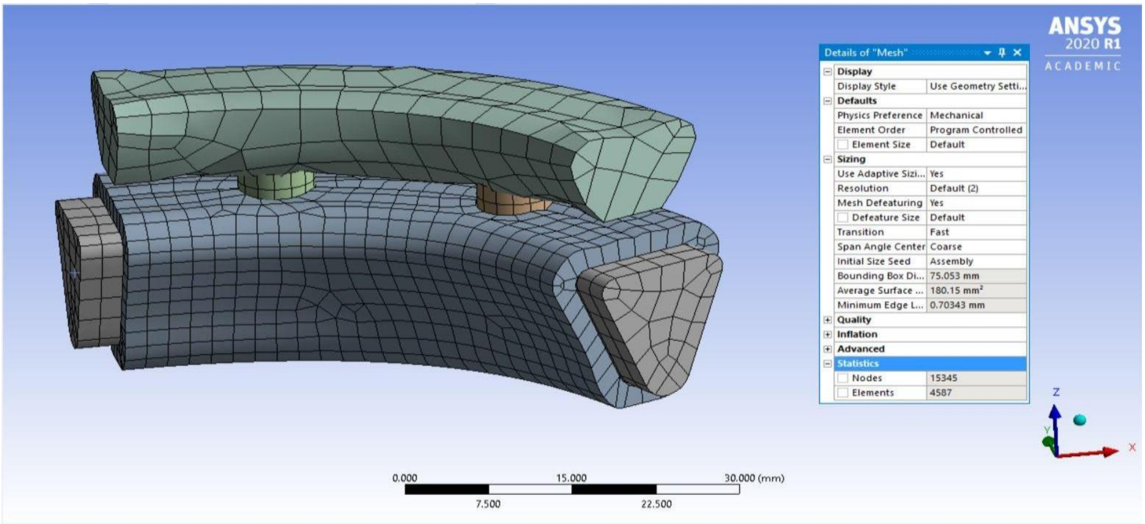

FIGURE 6: MESHING OF 3D MODEL

Application of fixed and forced constraint: In fixed constraint, bone assemblies (cortical and cancellous bone) were fixed so that they remain static during the application of forces. Following figure shows fixed constraint applied over bones:

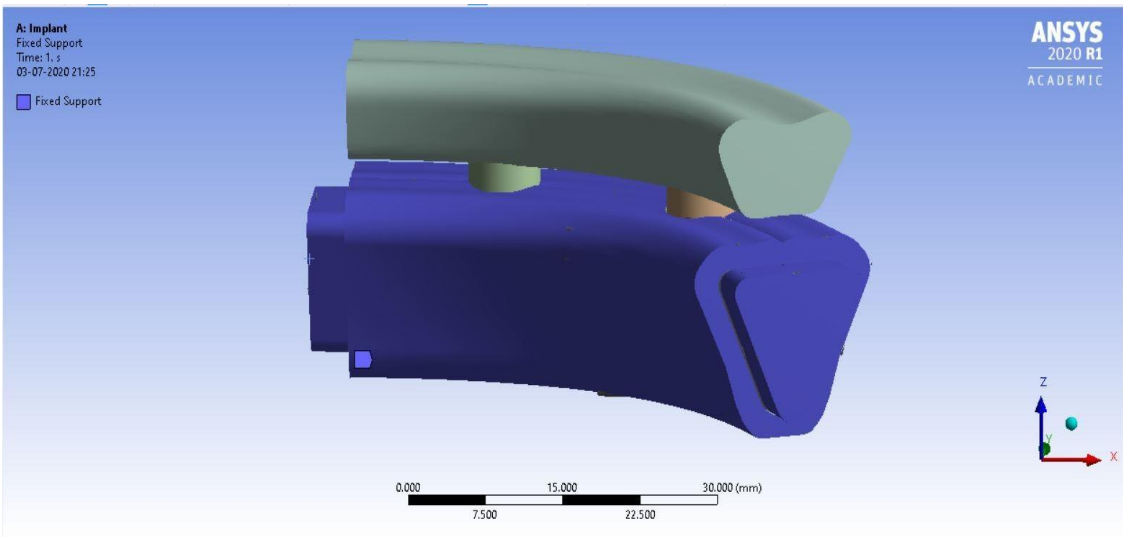

FIGURE 7: APPLICATION OF FIXED CONSTRAINTS

For forced constraint, forces were applied on the bridge and directions of forces were specified. Tangential and Compressive forces were applied to the bridge structure with a gradual increase in the magnitude. Four different magnitudes of 250 N, 500 N, 750 N, and 1000 N were selected which lie within the range of human bite force capability.

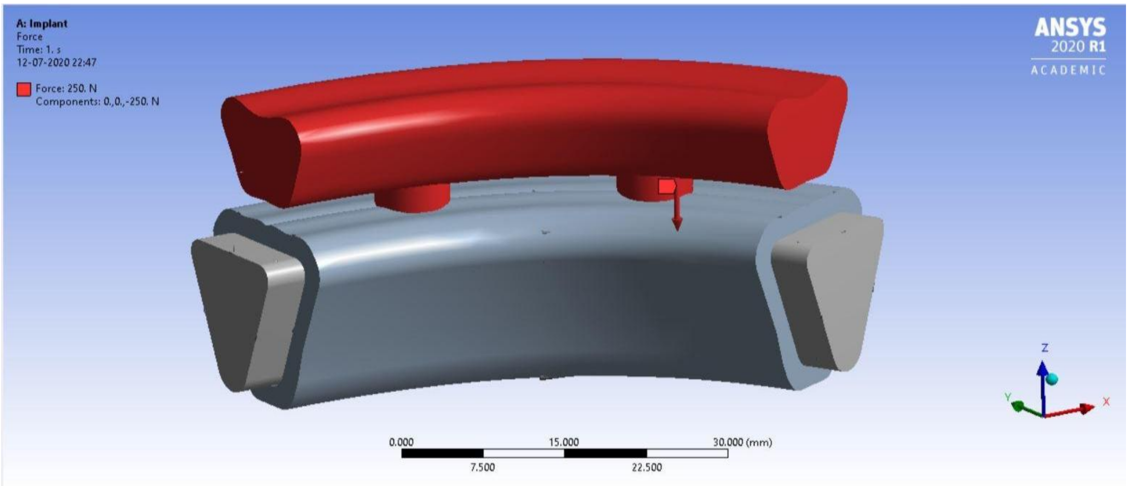

FIGURE 8: APPLICATION OF COMPRESSIVE FORCE

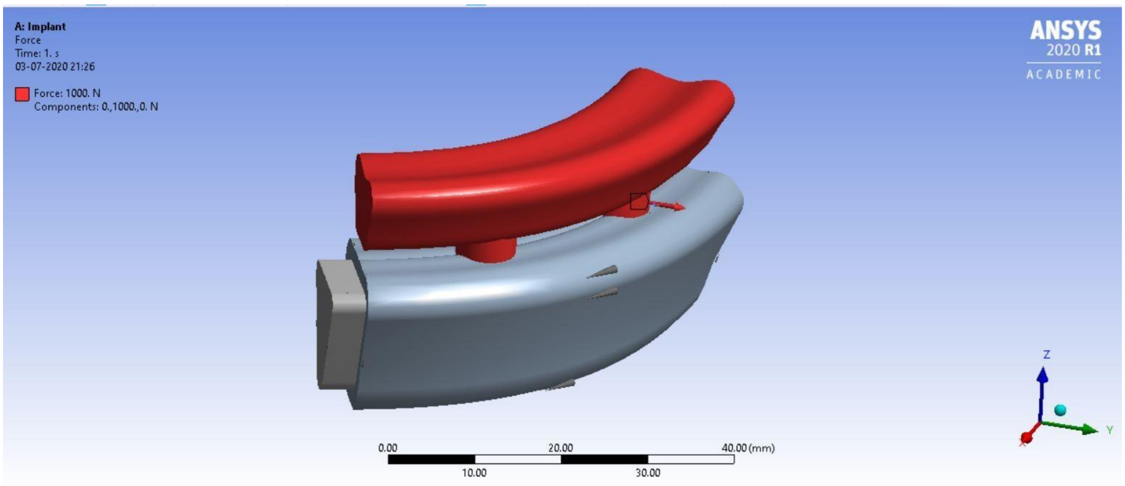

FIGURE 9: APPLICATION OF TANGENTIAL FORCE

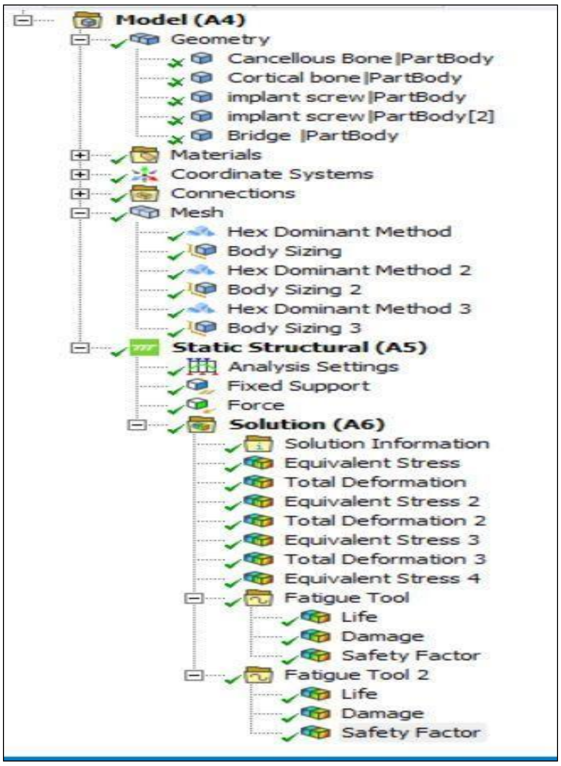

FIGURE 10: APPLYING FATIGUE TOOL

Fatigue Analysis of Implants: Dental Implants are subjected to cyclic forces during their day-to-day operations. These cyclic forces create fatigue stress within the implant body and the outer surface of the cortical and cancellous bone which is in contact with the implant. In fatigue stresses, failure occurs in the three main steps which are crack initiation, crack propagation, and rupture. The dynamics of the component and its location impact the fatigue stress produced in the dental assembly. Hence, time history of stress and strain at the exact location is an important factor in fatigue analysis. The surfaces subjected to fatigue load should be finely meshed to obtain accurate results. Finite Element Analysis software calculates the maximum and minimum implant and bone life. [6] The minimum life represents the time period between the application of the first cyclic force and appearance of the first crack within the body under consideration, whereas maximum life represents the total failure of the implant and bone.

Fatigue life calculation in ANSYS: For calculation of Fatigue life following process was carried out in ANSYS.

- Fatigue Tool was selected from the solution menu for life calculation.
- All the required material properties were defined as an input.
- For the output results, life damage and safety options were selected.
- After defining force magnitude and number of cycles, results were obtained.

Below is the flowchart for the methodology used.

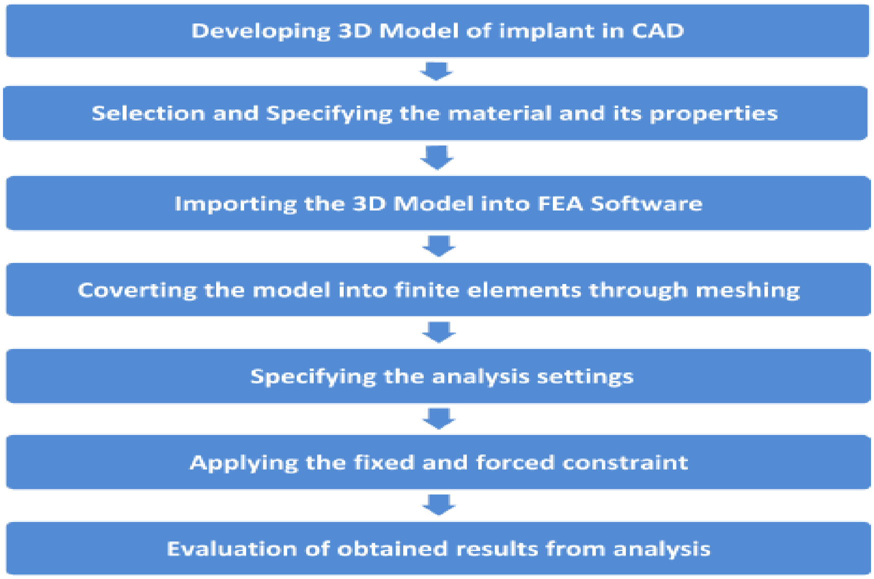

FIGURE 11: APPLYING FATIGUE TOOL

RESULT

When forces are externally applied on the bridge forces are developed within the body which tends to resist the applied forces. In this way, stresses are induced within the body. Due to stresses, the implant body starts to deform and when the value of stresses exceeds the permissible limit it starts to undergo failure. The stress values are obtained through numerically calculated finite element analysis and implant life is dependent on the magnitude of stresses induced. The following figures show the deformation caused by applied forces and stress distribution within various parts of the dental assembly.

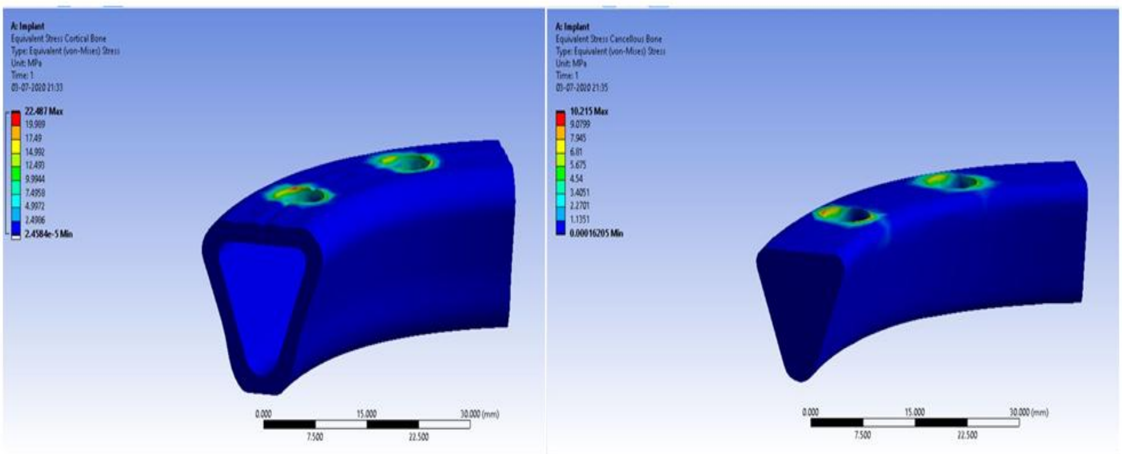

FIGURE 12: STRESSES INDUCED IN CORTICAL AND CANCELLOUS BONE

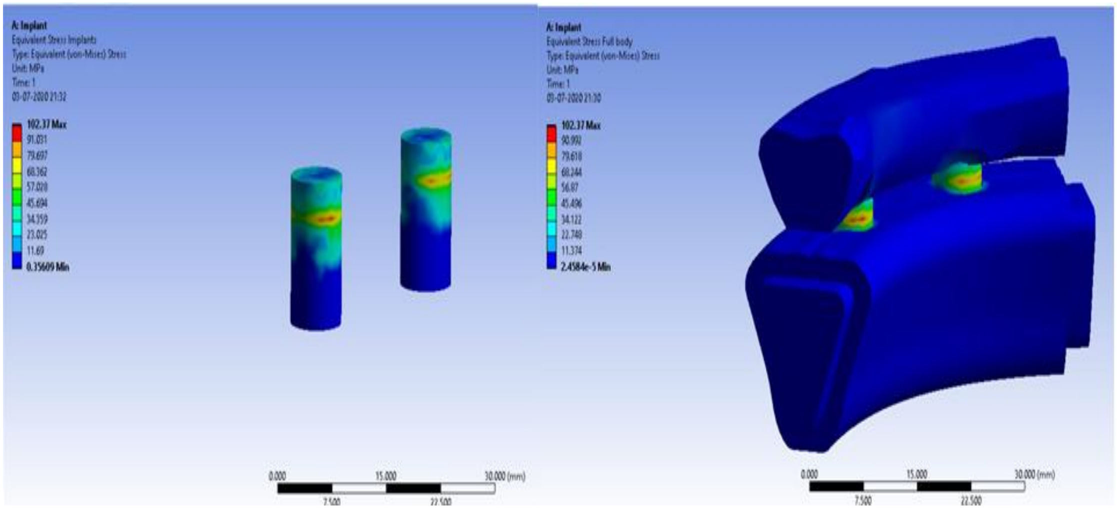

FIGURE 13: STRESSES INDUCED IN IMPLANTS AND DENTAL ASSEMBLY

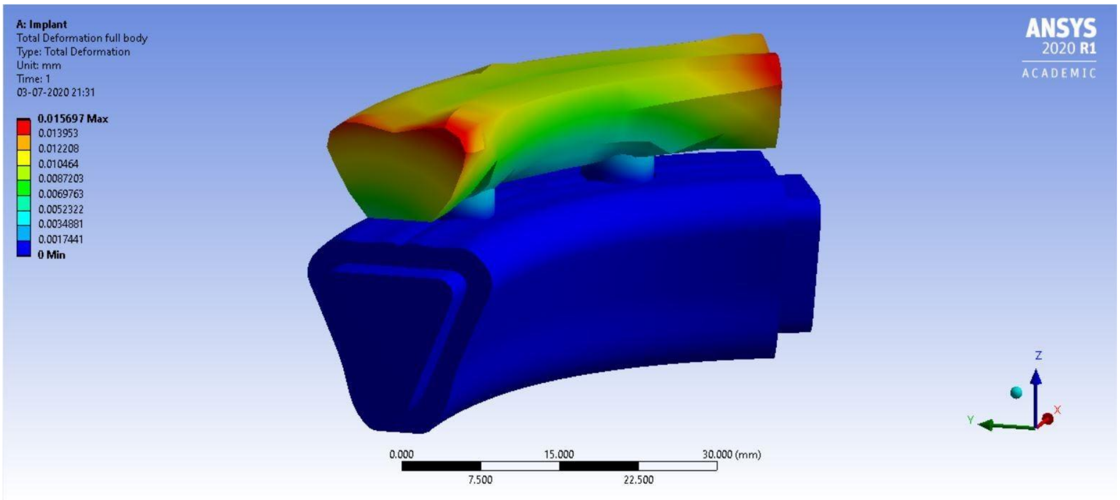

FIGURE 14: TOTAL DEFORMATION OF DENTAL ASSEMBLY

Implant Life:

Considering 400 cycles per day forces with a gradual increase in magnitude were applied to the bridge of dental assembly and implant life was calculated. Following figures show the implant life calculated from the finite element analysis:

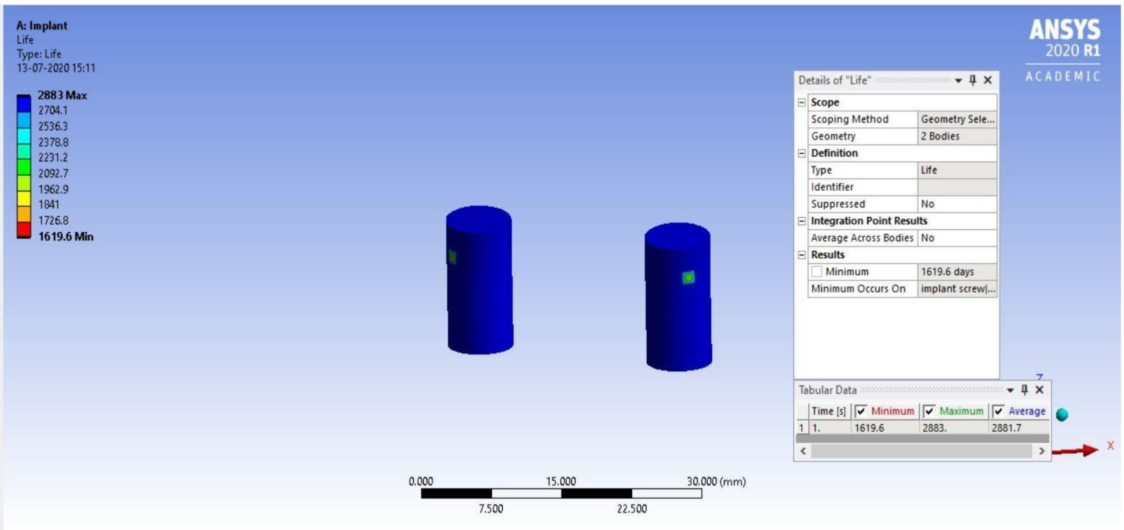

FIGURE 15: IMPLANT LIFE FOR 250 N.

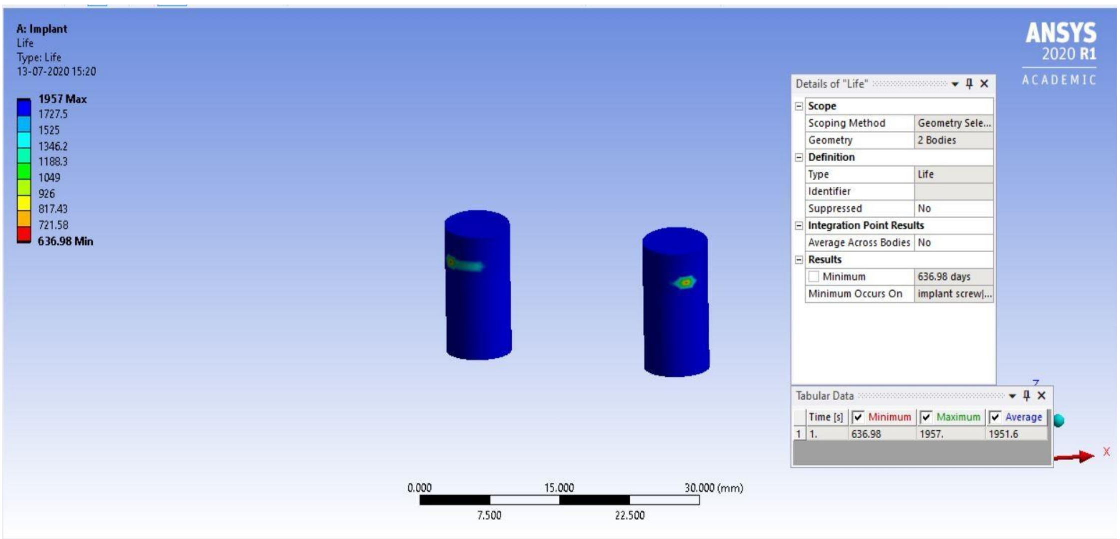

FIGURE 16: IMPLANT LIFE FOR 500 N

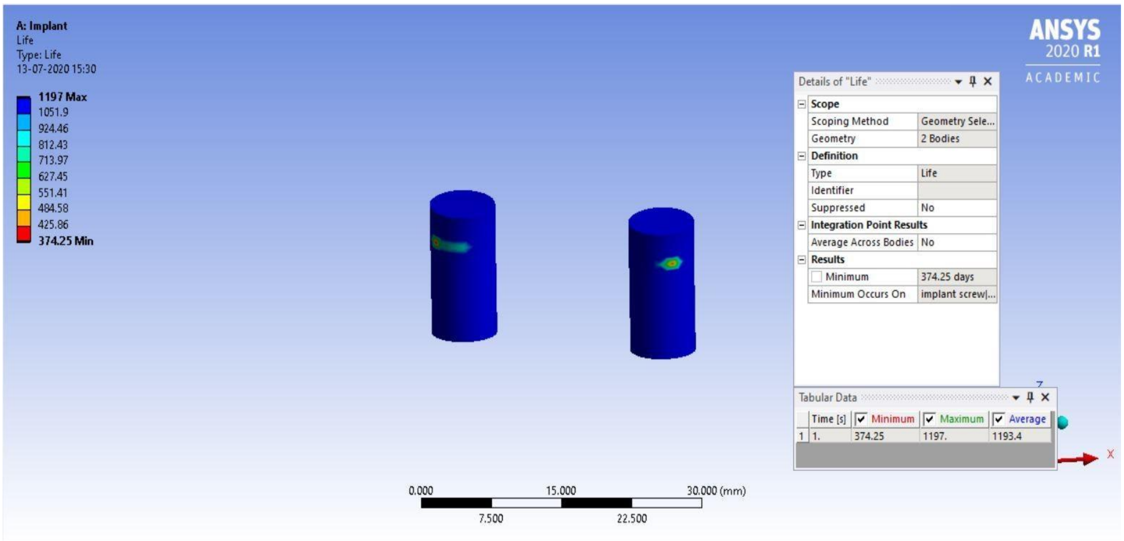

FIGURE 17: IMPLANT LIFE FOR 750 N

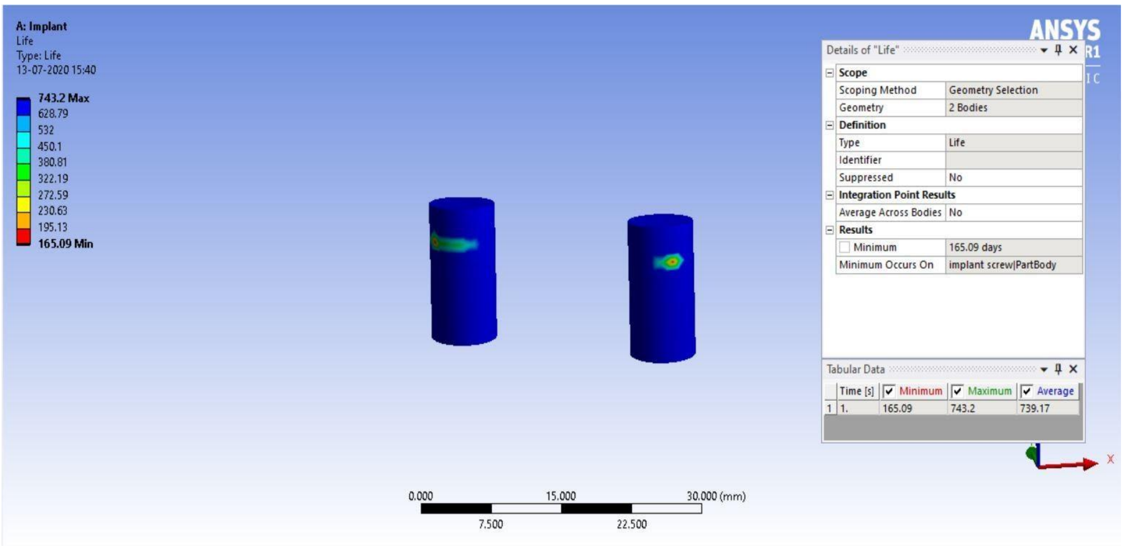

FIGURE 18: IMPLANT LIFE FOR 1000 N

Applied Force (in Newton)	Implant Life (in Days)		
	Minimum	Average	Maximum
250N	1619	2881	2883
500N	637	1951	1957
750N	374	1193	1197
1000N	165	739	745

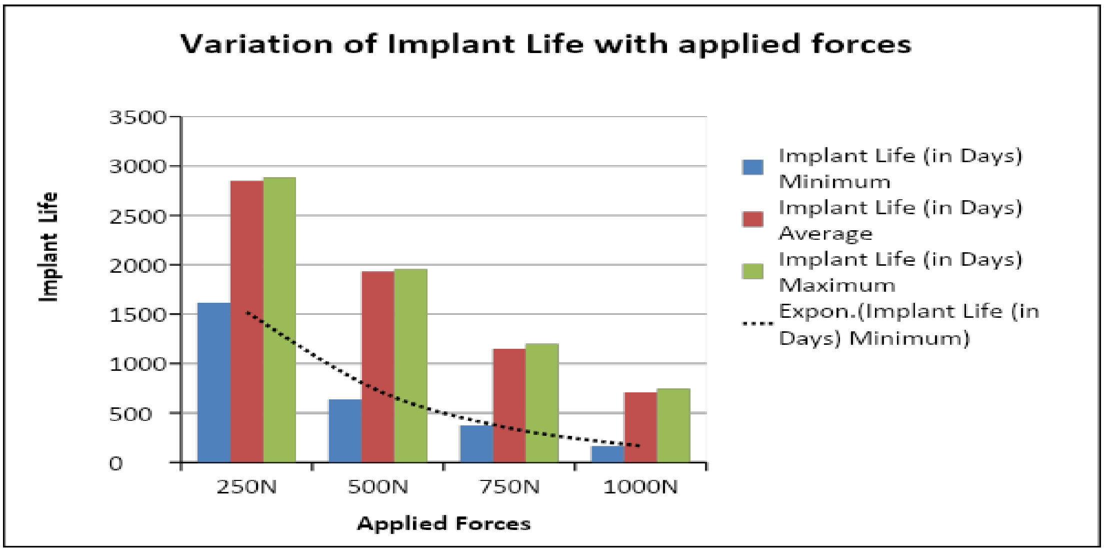

FIGURE 19: IMPLANT LIFE VS. APPLIED FORCES GRAPH

DISCUSSION

Since the invention of dental implants in the 1960s by Brånemark, dental implants have been a vital treatment option for the missing teeth in dental practice. Advances in technology have revolutionized the use of implants for the missing teeth in daily dental practice. But the survival of implants depends upon various factors like age, length and diameter of the implant, quality of the bone, pre-existing co-morbid conditions of the patients, and bite forces. Even though the success rates of implants are high, failure of implants is a nightmare for every dentist. Implants may fail during the healing or the functional phase. Inflammation is the main reason for failure during the healing phase. Failure of the functional phase is mainly because of the incorrect orientation of load on the long axis of the implant which can further affect the occlusion, masticatory force, and the quality of the adjacent bone. [9]

The various forces which act on the implants vary in their magnitude, direction, and type of stress they may induce in the implant. So it has become important to analyze the impact of these forces on implant life. [5] In order to predict the behaviour of the implant to applied force along with stresses and deformation-induced, finite element analysis serves as the most reliable and cost-effective technique. It also gives an approximate prediction of the implant life which can be further used to improve the implant design and bridge structure, thereby increasing the overall life of the implant. [4]

Finite Element Analysis is the process of simulation of physical models through numerical techniques for determining how the model will behave when put to use in a real-life environment. Through the application of finite element analysis, the number of physical prototypes and experiments can be reduced. This method is used in the design phase to optimize the products and to increase their functionality. Initially through the application of FEM difficulties in the structural analysis were solved, today it has wide application in various fields including medical applications as well. As the model is divided into a finite number of parts called elements, the accuracy of the simulation depends on planning, type, the total number of nodes used. [2]

For the prevention of implant failure caused by mechanical parameters, they should be evaluated well in advance. There are many traditional ways for analyzing these mechanical factors such as photo elastic model analysis, strain gauge analysis, which include making costly prototypes, applying real world forces, and calculating the stresses and deformation by manual methods. [1] But they have limitations such as high time and cost consumption along with unreliability of results. The use of FEA overcomes all of these limitations and also helps to improve the survival rate or life of the implants during the design phase. [2]

Application of FEA in implant dentistry includes in depth qualitative examination of the implant and its relation with the tooth, cortical, and cancellous bone. From the FEA analysis report, stress distribution and deformation in implants and bone can be calculated which correspond to results obtained from traditional methods. [1]

Advantages of FEA:

- It is a non-invasive technique.
- The FEA process can be repeated several times with ease.
- The time required for the analysis is less as compared to traditional methods.
- Specification and the material properties of anatomic craniofacial structures can be evaluated.

Disadvantages of FEA:

- The use of incorrect input information will lead to totally misleading results.
- Skilled computer operator having software knowledge is required
- There is need to have detailed information about their mechanical behaviour
- Accuracy of the results depends on the people associated in the study

By using Finite Element Analysis, stresses and deformation formed on the bone by the implant can be predicted. The Goodman, Soderberg or Gerber's criteria can be then used for fatigue analysis of the implant. [14]

In the following study, the authors have predicted the life of an implant correlating with the stress and deformation caused by the forces applied to the implant. Finite element analysis and ANYS stimulation for this. First, a 3D model of the implant supported bridge was developed. The authors took the properties of the implant material into consideration. Titanium metal was used for the implant material used. The various properties of cortical, cancellous bone and the human teeth were also taken into consideration. The whole 3D assembly of the implant-supported bridge and the bone was divided into distant node points by a process called meshing. A hexahedron dominant type of meshing was used for this study as it sustains the mechanical stiffness. The implant bone assembly was fixed during the application of forces. Tangential and compressive forces of four different magnitudes [250 N, 500 N, 750 N, 1000 N] were applied.

As the implant is subjected to the cyclic forces daily, the cortical and cancellous bones bear the fatigue stress. Dynamics of the implant component and the time history of forces play a major role in fatigue analysis. By using the ANYS stimulation, fatigue analysis was carried out. Goodman's formula was used for our study. On application of the forces, the authors were able to conclude that the maximum stresses are generated at the abutment screw junction. The life of the implant diminished with constant increase in the force.

CONCLUSION

Within the limitations of this study, it can be concluded that the implant life exponentially decreased with the increase in the forces. Thus, Finite Element Analysis is a helpful tool to determine the mechanical properties of implants and predicting the life of implant on application of different forces. Even though there are limitations, finite element analysis' experiments are repeatable and the designs can be changes and modified as per requirement. It is a computerized study and the results can be acknowledged qualitatively. [11]

ACKNOWLEDGEMENT

We would like to express our sincere gratitude to Dr. Pragathi Bhat for her timely advice and motivation because of which this project was successful.

REFERENCES

1. Ali MH. Finite Element Analysis is A Powerful Approach to Predictive Manufacturing Parameters. Journal of University of Babylon for Engineering Sciences. 2018; 26(1):229-38.
2. Kueh JT, Faris T. Finite element analysis on the static and fatigue characteristics of composite multi-leaf spring. Journal of Zhejiang University SCIENCE A. 2012 Mar 1; 13(3):159-64.
3. Patil MV, Bhasme MS, Kandkar MA, Padale MS, Tashildar MS. A Review on Design and Analysis of Hydraulic Powerpack and Pumps.
4. Rubo JH, Capello Souza EA. Finite-element analysis of stress on dental implant prosthesis. Clinical implant dentistry and related research. 2010 Jun;12(2):105-13.
5. Gultekin BA, Gultekin P, Yalcin S. Application of finite element analysis in implant dentistry. Finite Element Analysis: New Trends and Developments. Rijeka, Croatia: InTech Publishing. 2012 Oct 10:21-54.
6. Reddy MS, Sundram R, Abdemagyd HA. Application of finite element model in implant dentistry: A systematic review. Journal of Pharmacy & Bioallied Sciences. 2019 May;11(Suppl 2):S85.
7. Waskewicz GA, Ostrowski JS, Parks VJ. Photoelastic Analysis of Stress Distribution Transmitted From a Fixed Prosthesis Attached to Osseointegrated Implants. International Journal of Oral & Maxillofacial Implants. 1994 Jul 1; 9(4).
8. Rotunno AF, Callari C, Froiio F. A finite element method for localized erosion in porous media with applications to backward piping in levees. International Journal for Numerical and Analytical Methods in Geomechanics. 2019 Jan; 43(1):293-316.
9. Cheng HC, Peng BY, Chen MS, Huang CF, Lin Y, Shen YK. Influence of deformation and stress between bone and implant from various bite forces by numerical simulation analysis. BioMed research international. 2017 May 28; 2017.
10. MacGinnis M, Chu H, Youssef G, Wu KW, Machado AW, Moon W. The effects of micro-implant assisted rapid palatal expansion (MARPE) on the nasomaxillary complex-a finite element method (FEM) analysis. Progress in orthodontics. 2014 Dec 1; 15(1):52.
11. Trivedi S. Finite element analysis: A boon to dentistry. Journal of oral biology and craniofacial research. 2014 Sep 1;4(3):200-3.
12. Reddy MS, Sundram R, Abdemagyd HA. Application of finite element model in implant dentistry: A systematic review. Journal of Pharmacy & Bioallied Sciences. 2019 May;11(Suppl 2):S85.
13. Mao J, Osborn JW. Direction of a bite force determines the pattern of activity in jaw-closing muscles. Journal of dental research. 1994 May;73(5):1112-20.
14. Agostinho Hernandez B, Paterno A, Capello Sousa EA, de Oliveira Freitas JP, Foschini CR. Fatigue analysis of dental prostheses by finite element method (FEM). InASME International Mechanical Engineering Congress and Exposition 2015 Nov 13 (Vol. 57380, p. V003T03A081). American Society of Mechanical Engineers.

How to cite this article: Desai SS, Kumbhar HN, Apoorva N et.al. Use of finite element analysis for predicting fatigue life of implant and its correlation with applied forces. International Journal of Science & Healthcare Research. 2020; 5(3): 423-433.

******