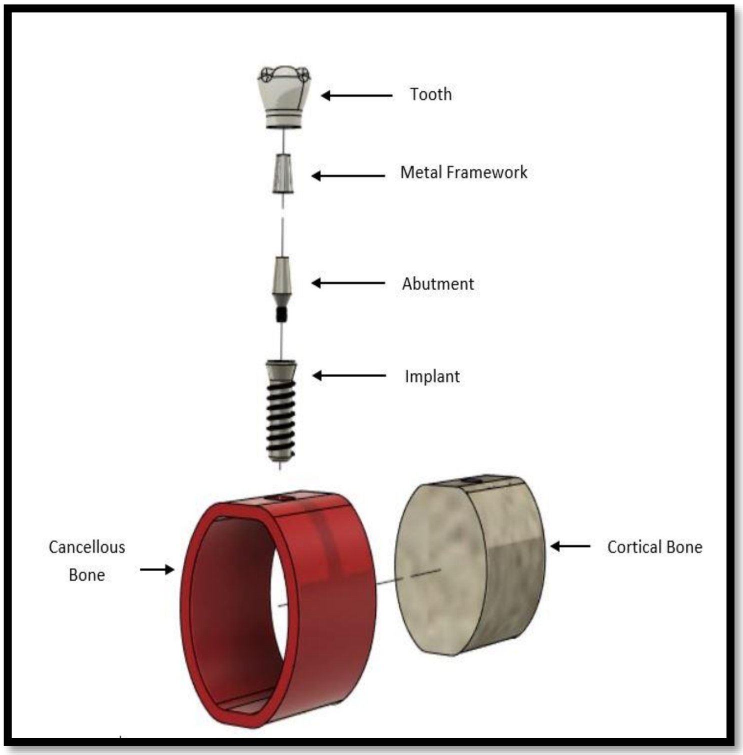

Static Behavior Of Dental Implant For Different Grades Of Titanium Alloys Using Finite Element Method

MEEM-5170

Submitted by,

Team 7

Rajat Ratnakar Gadhave

Sanket Kishor Kadam

Pratik Devendra Dalvi

Guided by: Dr. Susanta Ghosh

Table of Contents

1. Abstract

It is more important to develop and analyze relation among various implant structures, it's design and load distribution at bone-implant interface. This study proposes finite element analysis and stability under varying conditions of commercially available dental implant and implant bone interface. Literature survey for this study reveals that most of them have done static loading analysis, here we have represented influence of different grades of titanium alloys under static loading. Masticatory force is simulated by taking average of axial, lingual and mesiodistal direction force. In Ansys while performing Static Structural analysis frictional contact has been considered between implant-cortical bone, abutment-implant. In Ansys physical interaction between other parts is represented through bonded surface contact.

2. Problem Definition

From the middle of twentieth century development of dental implants and its parts was began and till now its ongoing research topic in medical as well as engineering industry in order to achieve long term success of this treatment. The success of this treatment depends on many factors out of which interface between abutment and implant is one of the important factors. In this research authors have analyzed dental implant model which is modelled as per medical standard. Assembly of this implant model along with other important members is shown below in Figure 1. To demonstrate results they have used Ansys software where static analysis evaluated and compared with some reference values for instance yield strength of material. After evaluating these assembly models at various standards, it is observed the von mises stresses developed in models are well below the maximum allowed criterion hence models are assumed to be safe.

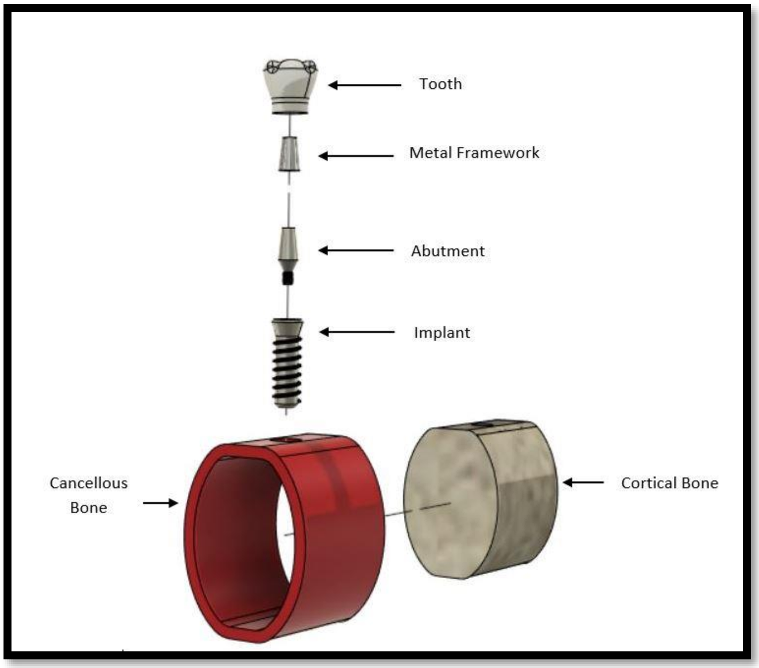

Figure 1. Dental Implant model with assembly

Temperature conditions assumed in this study shows standard behavior such as 15 ℃ for cold and 60 ℃ for hot. Also, it is assumed that these temperatures were maintained for period of one second in vicinity of model under consideration. In this study Ti–6Al–4V material was selected from Ansys material library for implant and abutment whereas metal framework was modelled by using cobalt-chromium alloy. Results of analysis for implant, abutment metal framework, cortical and spongy bone and occlusal surface material shows stress developed in this model were less than yield strength of respective materials.

3. Objective:

Objectives of this study are as follows:

1. To prepare Model of dental implant, abutment and other parts in the whole assembly.
2. Perform static analysis for different grades of titanium alloy on these modeled parts under Masticatory force conditions.
3. To compute factor of safety of different grade of Titanium used for implants.
4. To compute the stress results and factor of safety for Ti-6Al-4V and comment about the feasibility of the overall treatment.

4. Background:

As treatment of dental implant was introduced to facilitate for completely edentulous patient. The use of implants with modern analysis techniques resulted into remarkable long-term result. Reliability and stability of overall treatment mainly depends upon interface between each part. It is important to test this model under standard load conditions as excessive load can cause fatigue failure on the other hand underloading can also cause serious problem for this type of treatment. Important factor for success or failure of this treatment is also depend on stress distribution in entire bone structure. Main aim of this project is to test the dental implant of various titanium grade commercially available under static loading and finding its Factor of safety.

5. Approach:

5.1 Mesh Convergence:

Meshing is the most important step in the FEA analysis. The mesh quality of the CAD model is directly related to the accuracy of the analysis. Mesh subdivides the model into small elements over which the equations are solved. Thus, mesh refinement is required to obtain accurate results. In this project first a coarse mesh was generated. Coarse mesh requires less computation time and also the results are not accurate. But it gave rough verification to check the applied constraints and also where the maximum stresses are occurring in the system. As the geometry was complex the appropriate type of mesh element was SOLID 187, a 3D tetrahedral element. For mesh refinement first, the mesh span angle was changed from coarse to fine. After that further p-refinement was done, the element order was changed to quadratic which resulted in a 10 node Tetrahedral element. Then the h-refinement was carried out for the critical

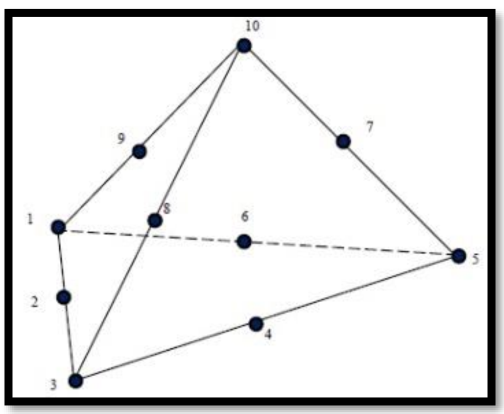

Figure 2. 10 node 3D Tetrahedral Element

parts such as the implant and the abutment where maximum stress was seen. The element size for the implant was gradually decreased from 0.7mm to 0.2mm and stress and deformation was tabulated as shown in table 1. After analyzing the results 0.45 mm element size was fixed for the implant as it gave close results to the paper referred. Then the same study was performed for abutment and element size of 0.3 mm was fixed for the abutment.

Element Size (mm)	Number of Nodes	Number of Elements	Von Mises Stress (MPa)	Deformation (mm)
0.2	534780	349212	395.34	0.014123
0.25	389064	246473	238.2	0.014354
0.3	353471	220717	352.88	0.014146
0.35	313536	192871	242.11	0.014527
0.4	285950	174335	253.58	0.014531
0.45	**268222**	**162247**	**211.01**	**0.01574**
0.5	257576	155134	216.55	0.016409
0.55	248421	149114	235.55	0.01614
0.6	274084	166469	244.37	0.01646
0.65	270361	164010	458.68	0.01873
0.7	251666	151448	260.9	0.18657

Table 1 . Mesh Refinement study for implant

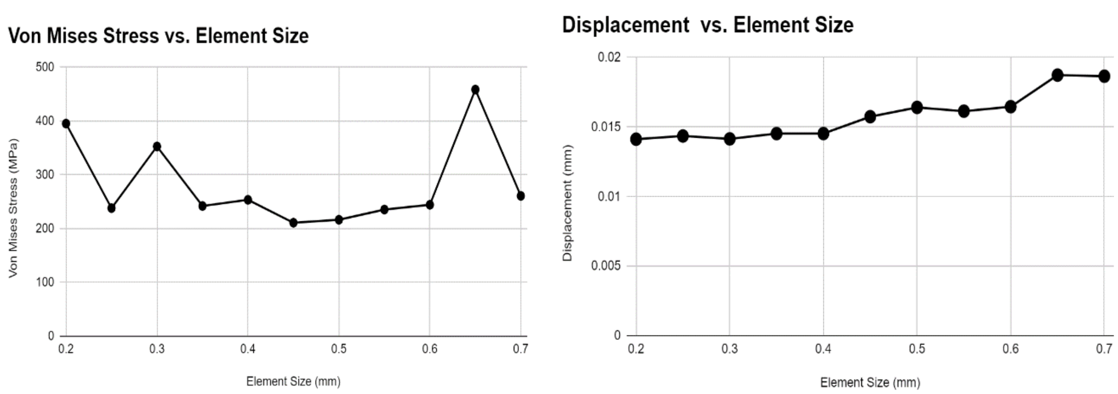

Figure 3. Von Mises stress and Displacement vs Element size

5.2 Mesh Details

The final mesh of an entire assembly which comprises of abutment, dental implant, metal framework, bone and occlusal part was made up of 162247 elements. The element used was a SOLID 187, a 10-node 3D tetrahedral element. Element size of 0.45mm for the implant and 0.3mm for the abutment and the other parts had an adaptive mesh which was used for the final analysis.

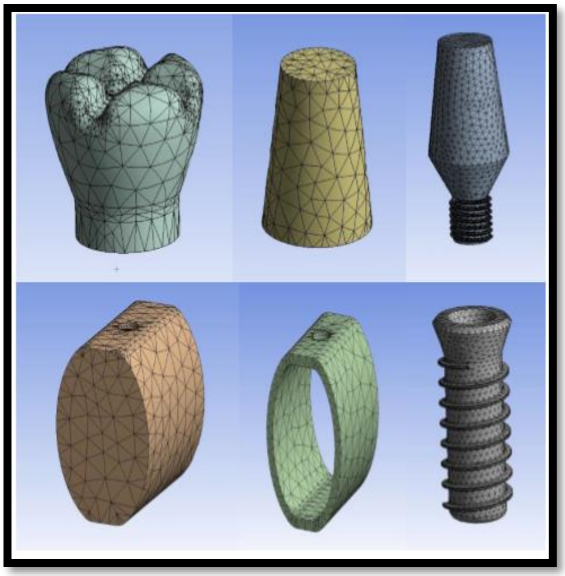

Figure 4. Final mesh of all parts

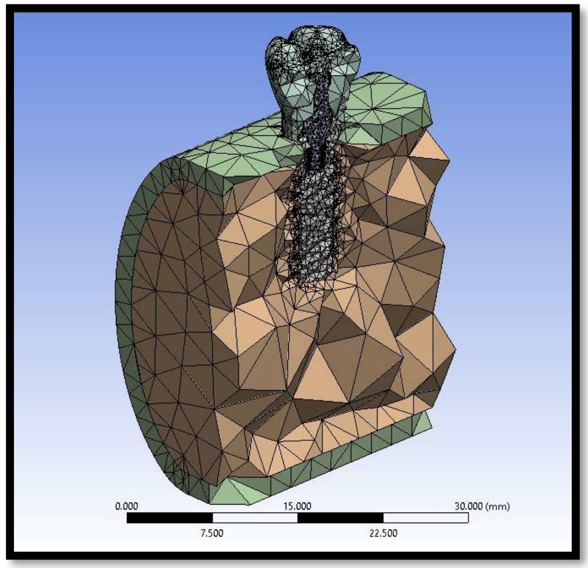

Figure 5. Mesh Model Assembly

5.3 Loading/Boundary Conditions:

In order to replicate standard force developed during chewing i.e. the masticatory which is 114.6 N in Axial direction, 23.4 N in Mesiodistal direction and 17.1 N in Lingual direction force was considered for this project for static analysis which has a resultant force of 118.2 N in the angle approximately 75º to the occlusal plane which was applied at center of the uppermost part i.e. occlusal material as shown in figure 4. Fixed support was given to entire outside wall of the cancellous bone as shown in figure 6.

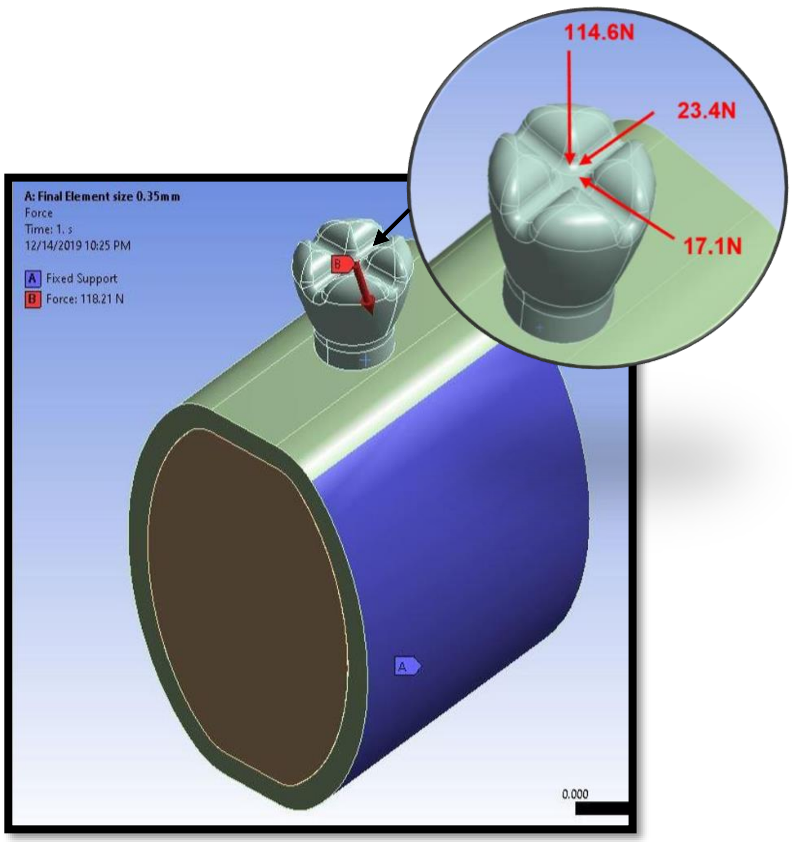

Figure 6. Assembly Under Loading

6. Results:

After performing the static analysis of the dental implant assembly, the maximum Equivalent Von Mises stress was found out to be 168.1 MPa and the total deformation to be 0.01574mm. The whole dental implant assembly had a factor of safety of 2.51.

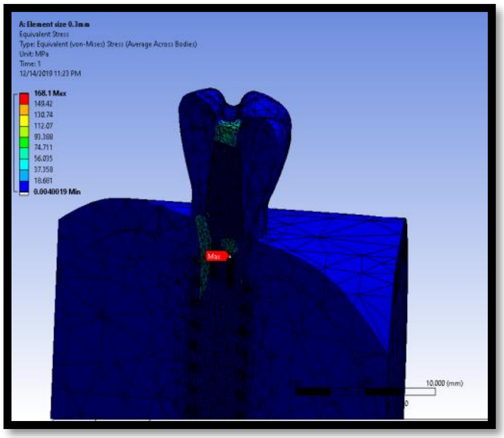

Figure 7. Stress Distribution in Dental Implant Assembly

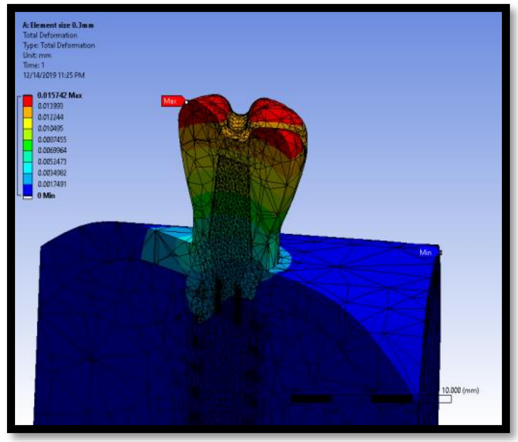

Figure 8. Displacement in Dental Implant Assembly

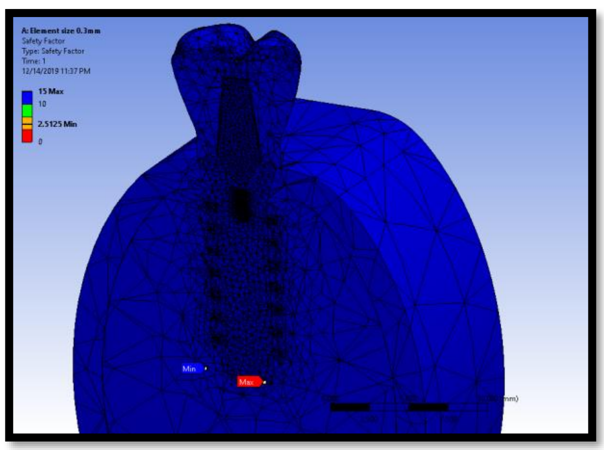

Figure 9. Factor of Safety in Dental Implant Assembly

Properties	Grade 1	Grade 2	Grade 3	Grade 4	Grade 5 (Ti-6Al-4V)	Ti-13Nb-13Zr
Tensile Strength (MPa)	240	345	450	550	1000	1030
Yield Strength (MPa)	170	275	380	485	800	900
Factor of Safety Comparisons						
Dental Implant	1.015	1.64	2.26	2.89	4.77	5.37
Abutment	1.01	1.63	2.26	2.88	4.75	5.34

Table 2. Results comparing different grades of Titanium Alloy Under Static Loading Condition

After performing the static analysis of the dental implant assembly, we found out the Maximum Von Mises stresses that occurred at the Dental Implant, Abutment, Metal Framework, Porcelain which is the tooth material and the two types of bone in jaws i.e. inner bone (Cortical Bone) and outer bone (Cancellous Bone). The results which we obtained from the static analysis are tabulated below in Table No. 2 for each part of the dental implant assembly.

Sr No.	Component Name	Static Loading	
		Equivalent (Von Mises) Stress (MPa)	Factor of Safety (FOS)
1	Dental Implant	167.44	4.77
2	Abutment	168.1	4.75
3	Metal Framework	22.77	15
4	Tooth (Porcelain)	83.75	5.96
5	Inner Bone (Cortical Bone)	51.74	2.51
6	Outer Bone (Cancellous Bone)	29.503	4.40

Table 3. Results of static analysis

6.1. Dental Implant:

Fig.10 represents the equivalent stress distribution in Dental Implant under Static analysis also the maximum induced stress in the implant is 167.44 MPa which is located by the probe in the Fig.10. The maximum stress is induced in the first thread where it is in contact with the abutment. Fig.11 represent the factor of safety of the Dental Implant for Static loading which comes out to be 4.77.

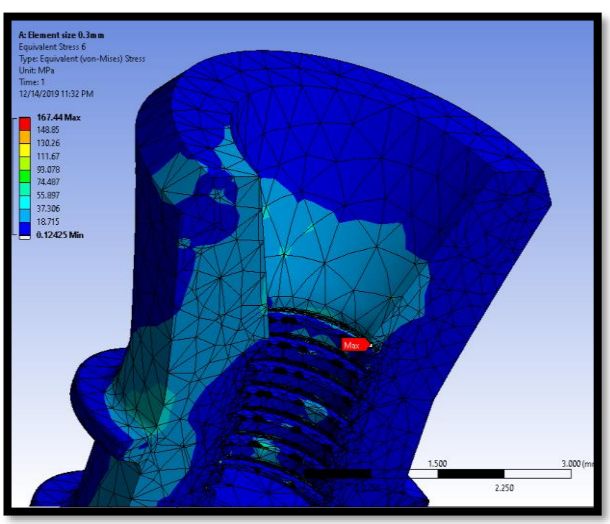

Figure 10. Stress induced in implant

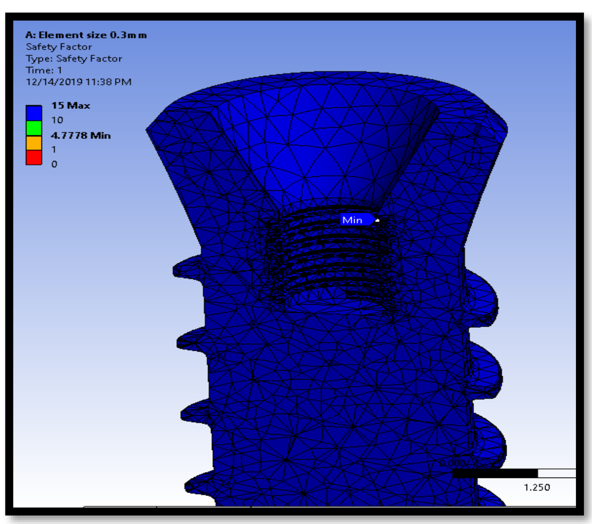

Figure 11. FOS for implant

6.2 Abutment:

Fig. 12 represents the equivalent stress distribution in Abutment under Static analysis also the maximum induced stress in the implant is 168.1 MPa which is located by the probe in the Fig.12. Fig.13 represent the factor of safety of the abutment for Static loading which comes out to be 4.75.

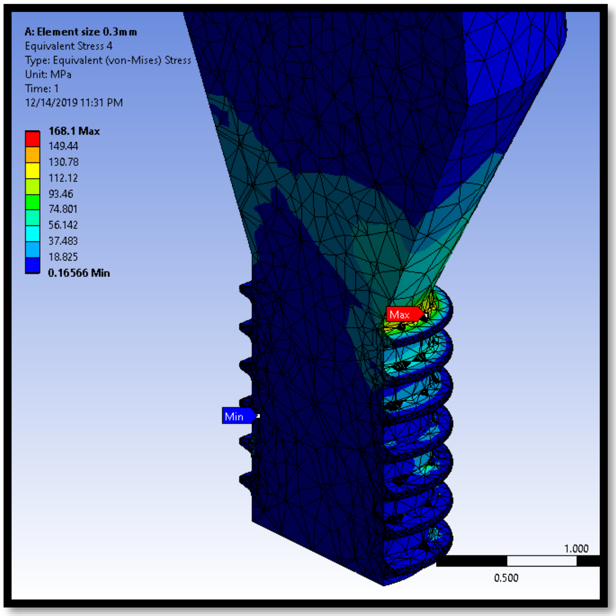

Figure 12. Stress induced in abutment

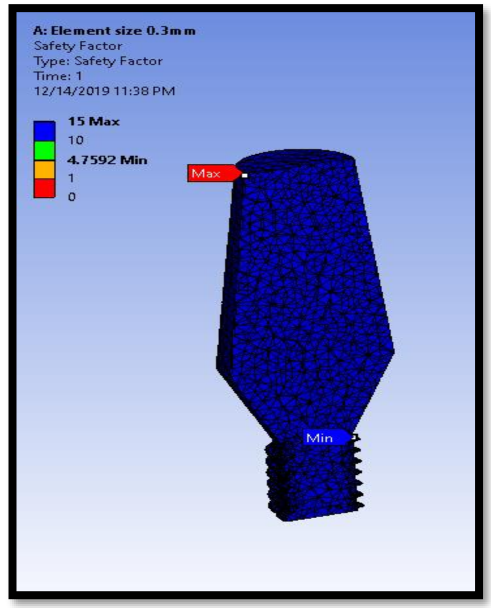

Figure 13. FOS for abutment

6.3 Metal Framework:

Fig.14 represents the equivalent stress distribution in Metal framework under Static analysis also the maximum induced stress in the implant is 22.77 MPa which is located by the probe in the Fig.14. Fig.15 represent the factor of safety of the abutment for Static loading which comes out to be 15.

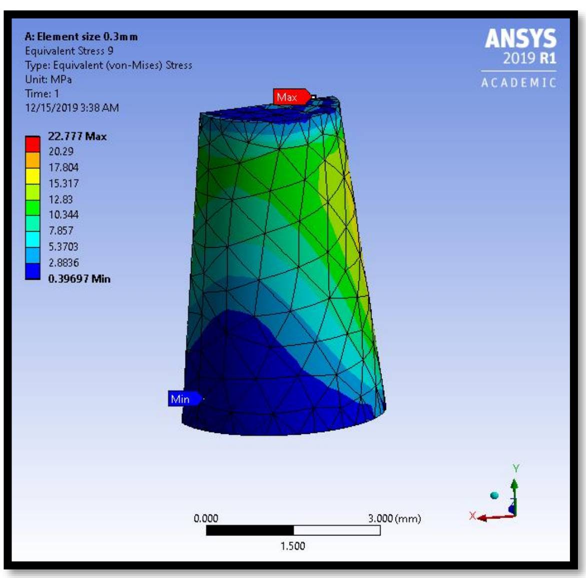

Figure 14. Stress induced in Metal Framework

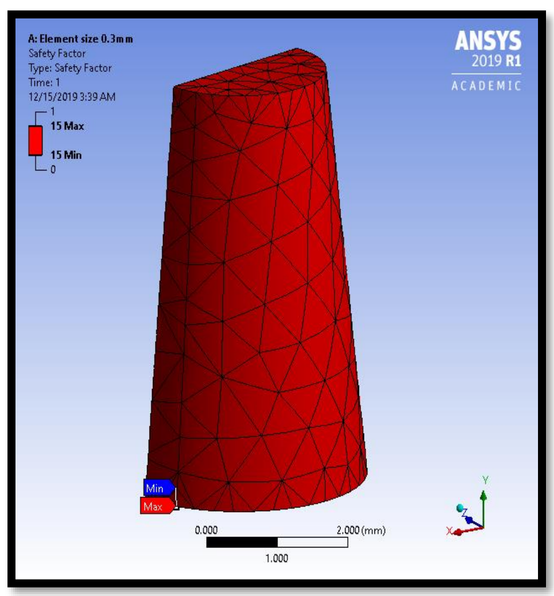

Figure 15. FOS for Metal Framework

6.4 Tooth (Porcelain):

Fig.16 represents the equivalent stress distribution in Tooth under Static analysis also the maximum induced stress in the implant is 83.75 MPa which is located by the probe in the Fig.16. Fig.17 represent the factor of safety of the abutment for Static loading which comes out to be 5.96.

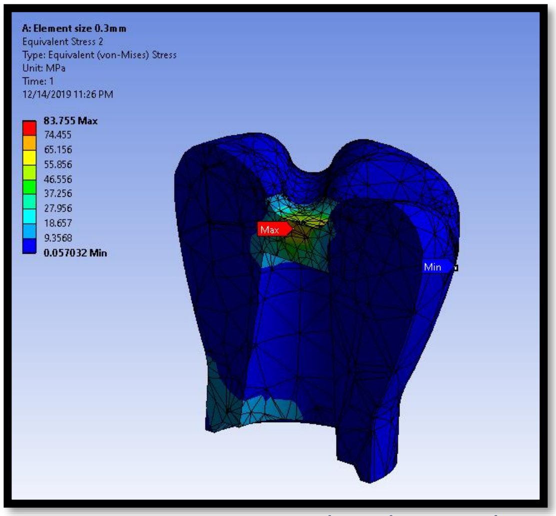

Figure 16. Stress induced in Tooth

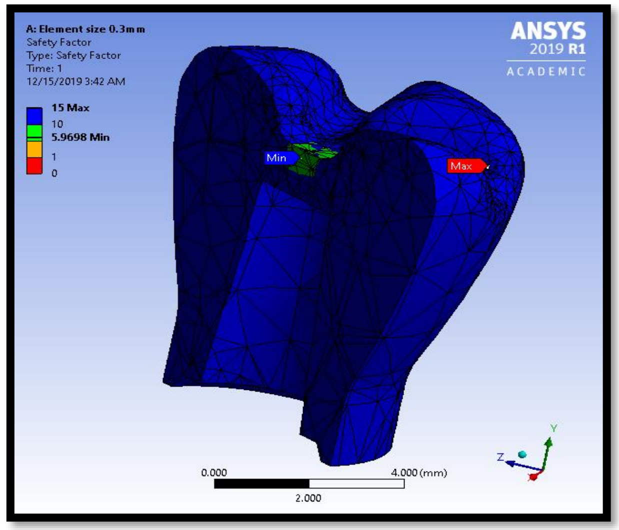

Figure 17. FOS for Tooth

6.5 Inner Bone (Cortical Bone)

Fig.18 represents the equivalent stress distribution in Cortical Bone under Static analysis also the maximum induced stress in the implant is 51.74 MPa which is located by the probe in the Fig.18. Fig.19 represent the factor of safety of the abutment for Static loading which comes out to be 2.5.

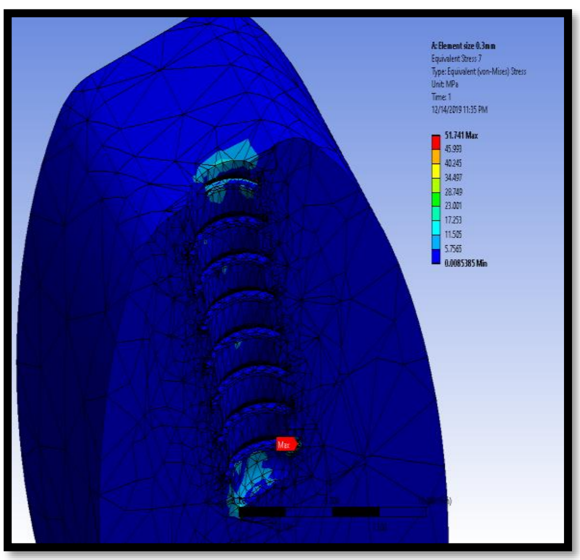

Figure 18. Stress induced in cortical bone

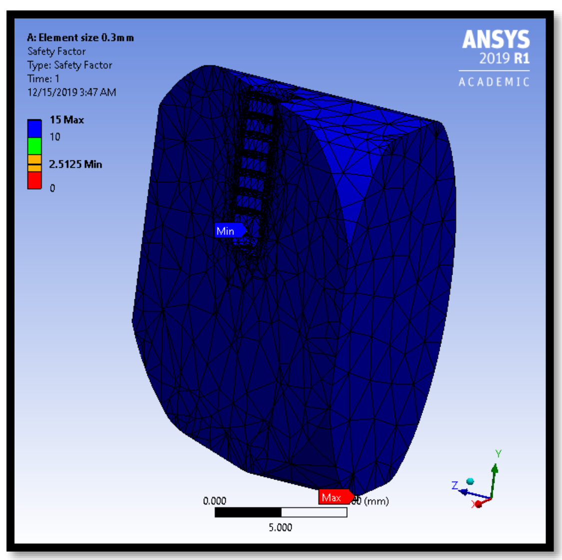

Figure 19. FOS for Cortical Bone

6.6 Outer Bone (Cancellous Bone)

Fig.20 represents the equivalent stress distribution in Cancellous Bone under Static analysis also the maximum induced stress in the implant is 29.50 MPa which is located by the probe in the Fig.20. Fig.21 represent the factor of safety of the abutment for Static loading which comes out to be 4.40.

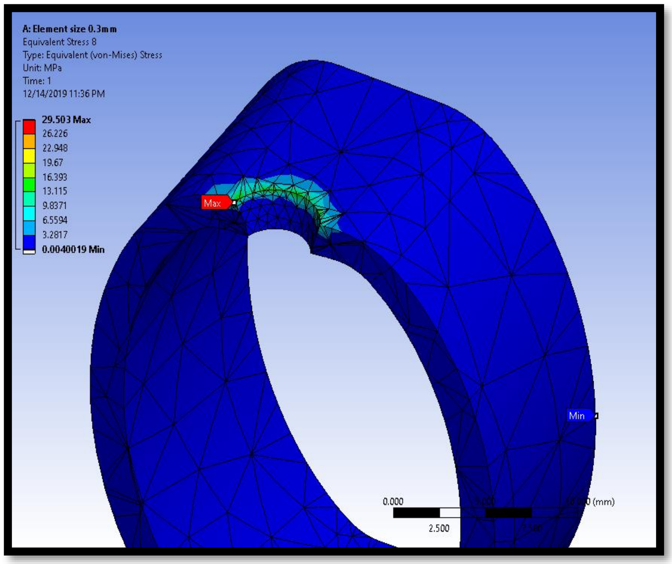

Figure 20. Stress induced in Cancellous Bone

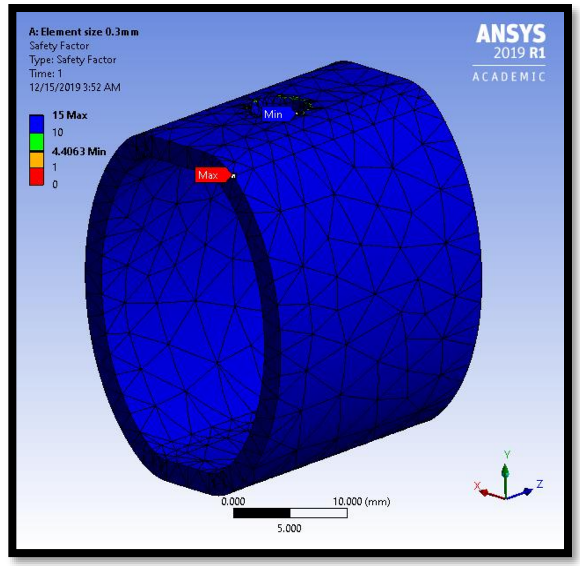

Figure 21. FOS for Cancellous Bone

7. Discussions

Finite Element Method is the most commonly use method to perform analysis in the field of Prosthetic and implant field to determine stress and fatigue and also for its design optimization.

For our analysis considering the scope and objectives of our project several assumptions were made in formulating the dental implant assembly model like we approximated the profile of implant thread and its helix angle from the Manufacturers catalogue. The material for the inner bone i.e. the Cortical bone is actually porous in structure but was assumed to be a solid (dense) part. Another assumption which we made was the cement layer which holds the metal framework with the abutment was not considered and was assumed to be bonded in nature. We found that the stress concentration is very specific to the region of the Dental Implant and Abutment. The research paper which we referred used SOLID45 types of element and we used SOLID187. We compared the values of Factor of Safety for different Grades of titanium and found that the Ti-13Nb-13Zr alloy has the highest FOS.

8. Conclusion

We were able to successfully perform the Static Behavior of dental implant and were able to compare our results with the research paper. We were able to determine the acceptable maximum stress induced in each component of the dental implant assembly and also were able to compute acceptable values of the factor of safety for each individual component and overall factor of safety for the complete assembly.

9. Future Scope

Now coming to the future scope, a lot of improvement can be made in modeling the dental implant assembly like mentioned earlier. Assuming the inner bone to be porous considering the cement layer between the metal framework and abutment. The pre-tensioning torque can be implemented to simulate actual clinical conditions. Fatigue analysis of the assembly can be performed if the S-N curves for the particular materials are obtained. The S-N curve would be the result of the actual testing of material under fatigue loading. Transient Thermal analysis can be performed by which we would be able to simulate the situations like consuming hot or cold food.

10. References

[1] O. Kayabaşi, E. Yüzbasioğlu, and F. Erzincanli, "Static, dynamic and fatigue behaviors of dental implant using finite element method," Advances in Engineering Software, vol. 37, no. 10, pp. 649–658, 2006.

[2] M. Mostafa and M. Tawfik, "Crack detection using mixed axial and bending natural frequencies in metallic Euler-Bernoulli beam," Latin American Journal of Solids and Structures, vol. 13, no. 9, pp. 1641–1657, 2016.

[3] Massachusetts Institute of Technology,Abaqus Documents, 2017. Retrieved from: https://abaqus-docs.mit.edu/2017/English/SIMACAEGSARefMap/simagsa-c-ctmmeshconverg.htm

[4] COMSOL Multiphysics, Finite Element Mesh Refinement, 2017. Retrieved From: https://www.comsol.com/multiphysics/mesh-refinement.

[5] Straumann Product Catalogue-2012

[6] Roland Masa, Gabor Barunitzer. (2017). Titanium and its alloys in dental implantology. National Center for Biotechnology Information.PMC,Article:PMC563369

Appendix

- CAD Models:

Figure 22. Tooth

Figure 22. Metal Framework

Figure 22. Abutment

Figure 22. Implant

Figure 22. Implant

Figure 22. Implant

- Material Properties:

Sr No.	Component Name	Material	Young's Modulus (GPa)	Poisson ratio	Yield Strength (MPa)
1	Dental Implant	Ti-6Al-4V	110	0.32	800
2	Abutment	Ti-6Al-4V	110	0.32	800
3	Metal Framework	Cobalt-Chromium alloy	220	0.30	720
4	Tooth	Feldspathic Porcelain	61.2	0.19	500
5	Inner Bone (Cortical Bone)	-	11.5	0.31	130
6	Outer Bone (Cancellous Bone)	-	2.13	0.3	130

Table 4. Material Properties

Ti-6Al-4V, Cobalt-Chromium alloy, Porcelain and Cancellous bone material have linear isotropic behavior. The cortical bone material is transversely isotropic.

- Contact Between Mating Surfaces

Cortical Bone with Cancellous Bone Bonded
Dental Implant with Abutment Frictional Contact (Coefficient of Friction 0.20)
Dental Implant with Cancellous Bone Frictional Contact (Coefficient of Friction 0.20)
Dental Implant with Cortical Bone Frictional Contact (Coefficient of Friction 0.20)
Tooth with Dental Implant Frictional Contact (Coefficient of Friction 0.20)
Metal Framework with Dental Implant Frictional Contact (Coefficient of Friction 0.20)
Metal Framework with Tooth Bonded
Metal Framework with Abutment Bonded
Tooth with Cancellous Bone No Separation