Journal of Harbin Engineering University
ISSN: 1006-7043

Vol 44 No. 11
November 2023

Investigating the Effect of Neck Length and Abutment Angle on Stress Distribution in Dental Implant Fixtures Using the Finite Element Method

Dana Rezaei[1], [2]Dr. Behzad Yathrebi

[1]Master of Medical Engineering-Biomechanics, Tabriz-Iran

[2]PH.D of Medical Engineering-Biomechanics, Tabriz-Iran

Abstract- Objective: The purpose of this research was to investigate the effect of neck length and abutment angle on stress distribution in dental implant fixtures using the finite element method. First, the K-Three fixture of Kimia Medical company with K.M.E brand and abutments with 0, 1 and 2 mm necks with 0, 15 and 25 degree angles were selected for study and then their modeling was done using Solid works software and then stress distribution analysis in The fixture of each designed model was done using Abaqus software.

Materials and methods: In this research, the effect of neck length and abutment angle on stress distribution in dental implant fixture was investigated by finite element method.

First, by using Solid works software, the desired models were simulated, and then, using Abacus software, the stress was checked based on the amount of standard forces applied in the cortical bone of the lower jaw.

Result: The analysis performed in Abaqus software showed that increasing the abutment angle from 0 to 25 degrees, the amount of stress increases by about 30% for the first 15 degrees and about 1% for the second 10 degrees, and as the length of the abutment neck increases from 0 mm to increases by 2 mm, the amount of stress decreases by 2.5% per mm.

It seems that the tension has a direct relationship with the abutment angle and an inverse relationship with the length of the abutment neck.

Keywords- dental implant - abutment neck length - abutment angles

Introduction

Tooth less ness has negative effects on people's appearance, chewing food and speech abilities [1], therefore, nowadays, dental implants are used as a replacement for lost teeth [2].

But considering that the material, design and dimensions of the parts used have a great impact on the forces applied to the jawbones and are effective in the life of the implant, so biomechanical research in this regard seems necessary [3].

• Advantages of dental implants

In 2018, Yuji Sato and colleagues proposed an approach to apply implant therapy to the elderly. This approach included using an appropriate type of implant considering the patient's remaining life expectancy and general condition, performing less invasive surgery, providing treatment using flexible prostheses that are easily manageable and can be modified, and managed oral hygiene by providing patients with an implant card when treatment was completed [4].

• The success rate of dental implants

In 2018, Souza and colleagues concluded after research that a total of 205 implants in patients with an average age of 64.1 years, the cumulative survival rate was 96.6% during a follow-up period of 37.3 months [5].

In 2018, after research, Lemos et al found that the average survival rate and implant success rate at the patient level (according to the number of patients) were 94.76% and 93.81%, respectively [6].

• Stress related to dental implants

In 2017, after research, Emil et al announced that the highest stress in cancellous bone was predicted when a titanium implant was used in the anterior maxilla, although the stress depends on the properties of the material and the characteristics of the model [7].

In 2020, after research, Shreya et al concluded that the main factors of bone loss are mechanical overload, abutment connection design, implant geometry, implant position, and bone density [8].

• The effect of changes in the components of dental implants

Mahmoudi et al. in 2018 (2019 AD) after research concluded that ideally implants should be placed parallel to each other, parallel to adjacent teeth and at the same height. However, in some cases, the anatomy and morphology of the remaining ridge prevent this from being achieved, and the implants must be placed at an angle [9]. Displacement of the abutment leads to the displacement of the tooth crown and as a result disrupts the act of chewing [10].

In 2018 (2019 AD), Hossein Netaj Miande and his colleagues concluded after research that increasing the vertical height of the crown and the ratio of the length of the crown to the implant reduces the stress on the abutment screw as the weakest member of the implant set, but it may increase The possibility of failure due to fatigue in the abutment and fixture as well as the loss of the surrounding bone [11].

• Implant implantation problems

Dental implants have been the first choice of patients to replace missing teeth due to its numerous advantages in preserving adjacent teeth and aesthetics, but may be more expensive than fixed bridges or removable prostheses [12].

Sometimes infection has been observed after the implant operation, and the symptoms continued despite antibacterial drug treatment and surgery, and by conducting research, implant treatment was rejected for patients with SAPHO syndrome [13].

Materials and methods

In this research, we modeled the K-Three fixtures of Kimia Kasht Ateba dental equipment company with the K.M.E brand, and these fixtures are made of grade 4 titanium. We also used the screws of this company for the screws, but the screws are made of grade 5 titanium, but since two colors were used to separate the straight and angled abutments, we also used both blue and gold colors in this research. We worked for the abutments, we used the abutments of this company, which are made of grade 5 titanium.

First, we modeled fixtures and abutments with necks of 0, 1, and 2 mm with angles of 0, 15, and 25 degrees for research, using Solid works software, and then analyzed the stress distribution in the fixture of each of the designed models using We checked the Abaqus software.

Fixture design

K-Three fixtures of Kimia Kasht Ateba dental equipment company with K.M.E brand are equipped with self-locking and they are made of grade 4 titanium. The final fixture model was obtained as follows:

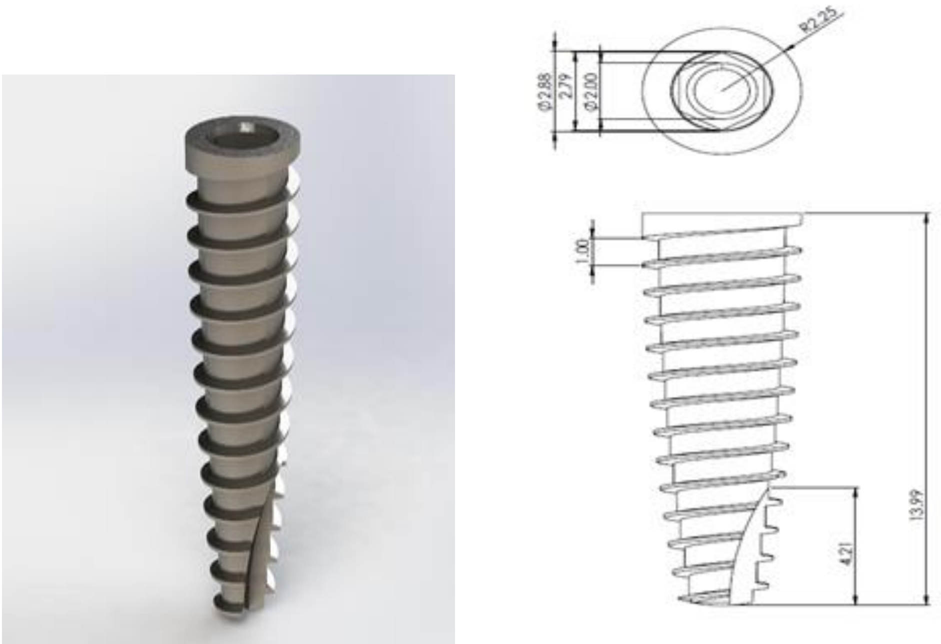


Figure 1) View of the designed fixture with its dimensions

Screw

Although the screws are made of grade 5 titanium, since we used blue and gold colors to separate the straight and angled abutments, we also used the same colors for their screws. The final screw model was obtained as follows:

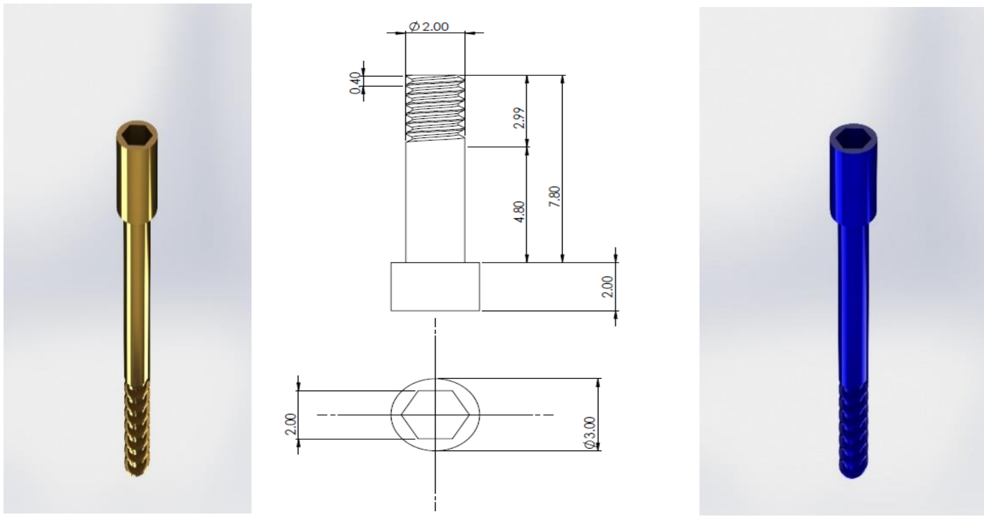


Figure 2) View of designed screws with their size

Abutment

Since our research includes straight and angled abutments, we designed both types in such a way that we designed an abutment with an angle of 0, 15, and 25 degrees and a neck length of 0, 1, and 2 mm. We designed 9 models, each of which will be described below:

1. Abutment with 0 degree angle and 0 mm neck length

These abutments are known as direct abutments and they are made of grade 5 titanium. The final model of this abutment was obtained as follows:

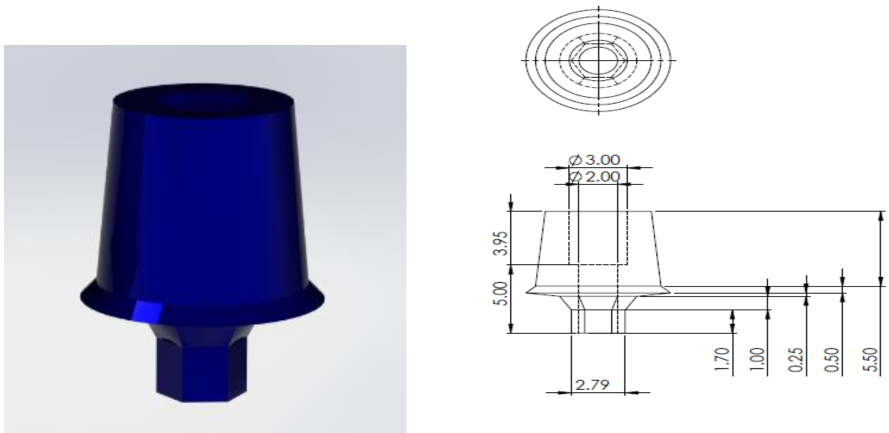


Figure 3) the final model of the abutment with a 0 degree angle and 0 mm neck length and its size

2. Abutment with 0 degree angle and neck length of 1 mm

The final model of this abutment was obtained as follows:

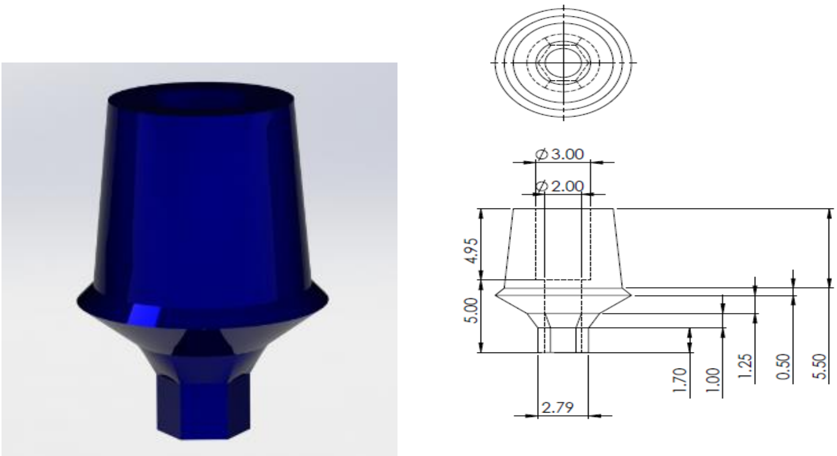


Figure 4) the final model of the abutment with a 0 degree angle and a neck length of 1 mm and its size

3. Abutment with 0 degree angle and neck length of 2 mm

The final model of this abutment was obtained as follows:

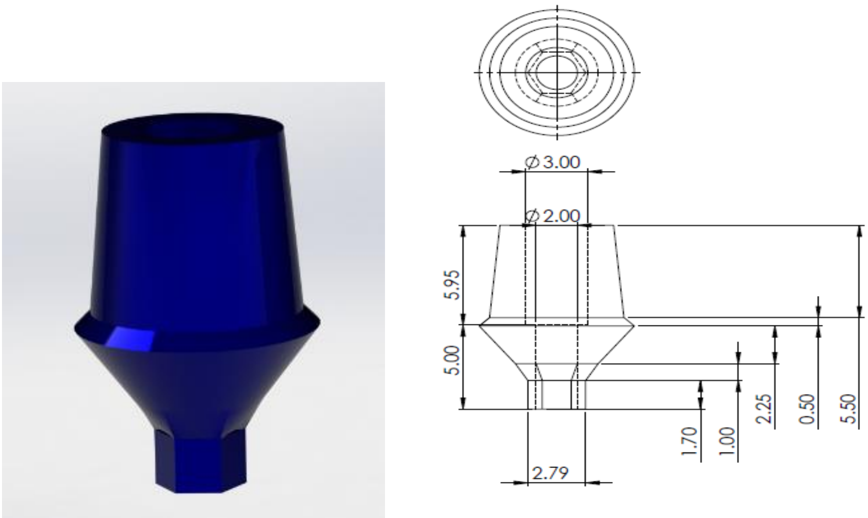


Figure 5) the final model of the abutment with a 0 degree angle and a neck length of 2 mm and its size

4. Abutment with an angle of 15 degrees and neck length of 0 mm

These abutments are known as angled abutments and they are made of grade 5 titanium. The final model of this abutment was obtained as follows:

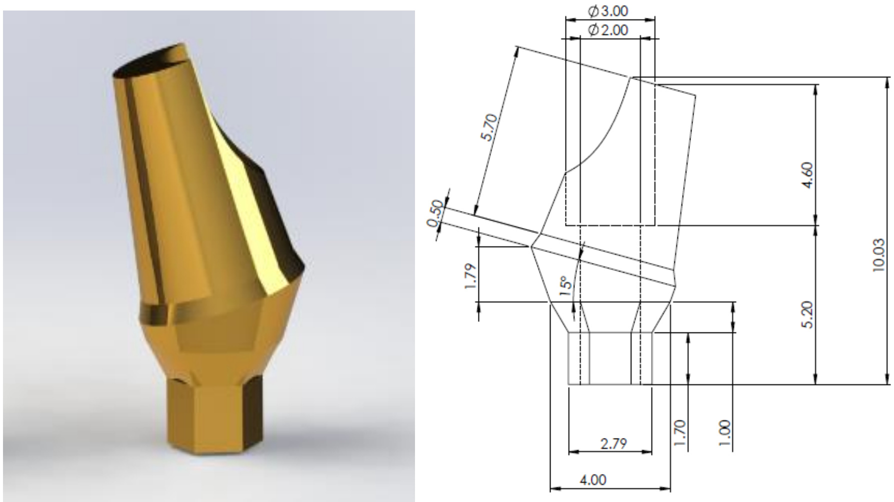


Figure 6) the final model of the abutment with a 15 degree angle and a neck length of 0 mm and its size

5. Abutment with an angle of 15 degrees and a neck length of 1 mm the final model of this abutment was obtained as follows:

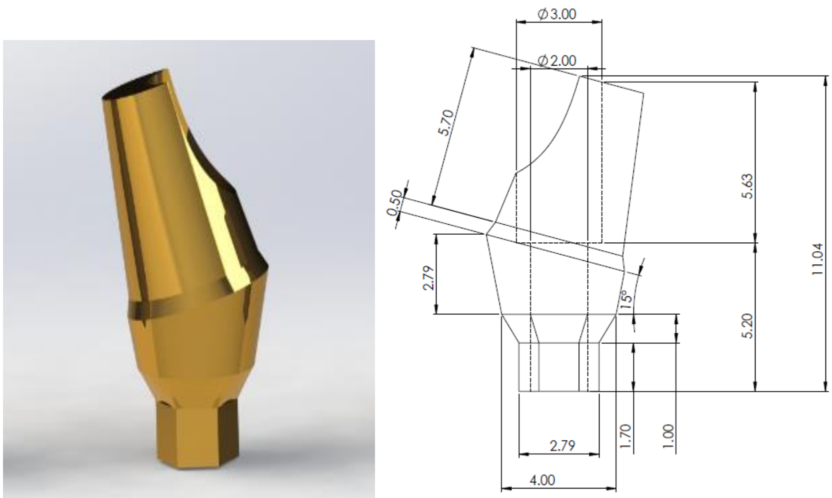


Figure 7) the final model of the abutment with a 15 degree angle and a neck length of 1 mm and its size

6. Abutment with an angle of 15 degrees and a neck length of 2 mm

The final model of this abutment was obtained as follows:

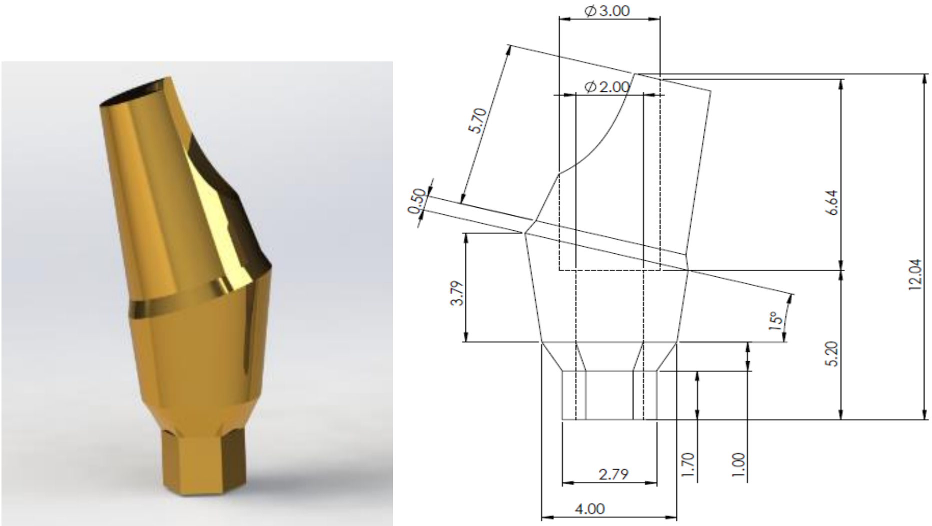


Figure 8) The final model of the abutment with a 15 degree angle and a neck length of 2 mm and its size

7. Abutment with an angle of 25 degrees and neck length of 0 mm the final model of this abutment was obtained as follows:

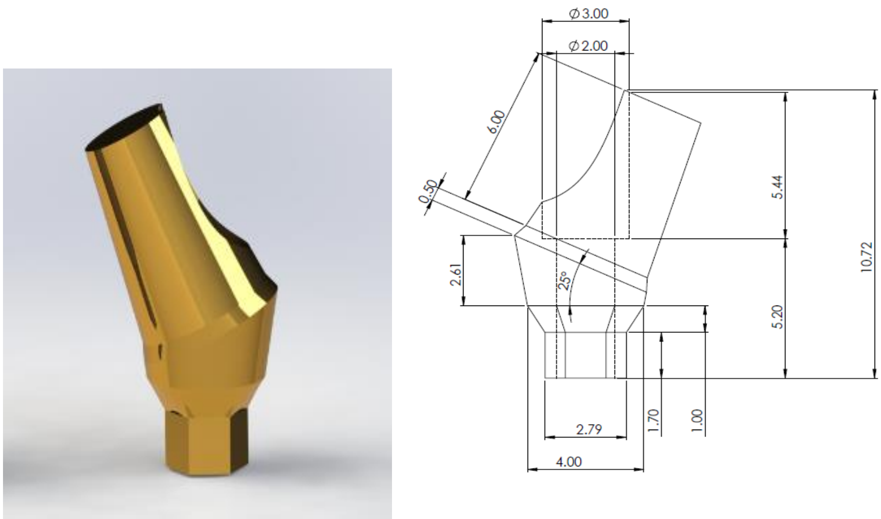


Figure 9) the final model of the abutment with a 25 degree angle and 0 mm neck length and its size

8. Abutment with an angle of 25 degrees and a neck length of 1 mm

The final model of this abutment was obtained as follows:

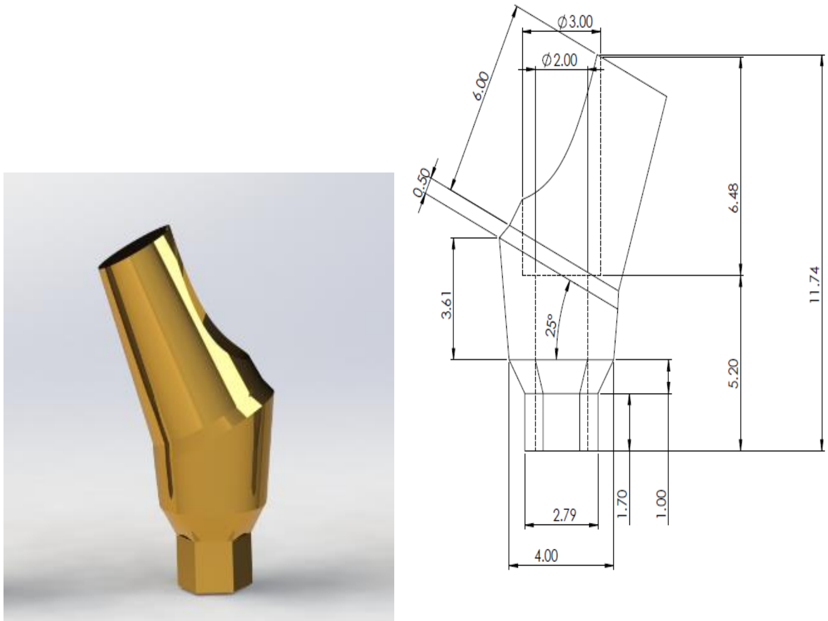


Figure 10) the final model of the abutment with a 25 degree angle and a neck length of 1 mm and its size

9. Abutment with an angle of 25 degrees and neck length of 2 mm

The final model of this abutment was obtained as follows:

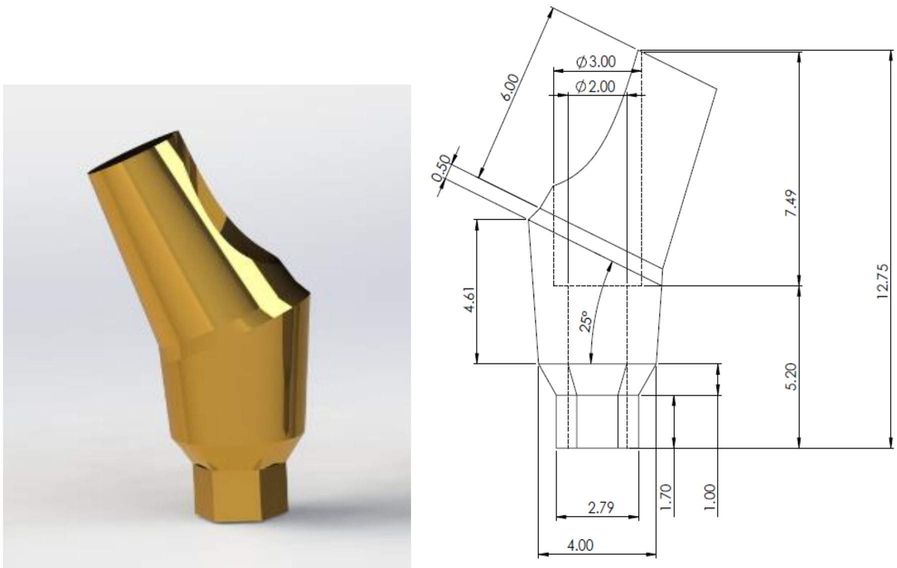


Figure 11) the final model of the abutment with a 25 degree angle and a neck length of 2 mm and its size

Analysis:

Model elementing: performing a finite element analysis and reaching the correct answers depends to a great extent on the correct meshing of the elements in this environment. In order to determine the size of the elements as well as the method of distribution and dispersion of the elements on an object, the granulation operation must be done on it. In the element section, there are various features for elementing a part. One of the main steps in elementing a model is choosing the shape of the element and the technique of elementing. According to the geometric shape of the model, the best type of element used is the quadrilateral type. Which has high accuracy compared to the triangular element.

One of the most important parameters in solving finite elements is the solution time discussion. This parameter directly depends on the number of elements created. If, for example, 100 elements are selected for gridding a problem, it will definitely take less time to solve the same problem with 200 elements. You may think that solving the problem with 100 elements is a more appropriate option, but the matter is a bit more technical and the issue is convergence in the answers. The number of elements, along with the geometry and apparent order of the element, are effective parameters in the discussion of the convergence of answers in Abaqus. You should use a reasonable number of elements at first. One of the essential points in determining the initial number of elements is the discussion of geometry and the grid order.

After proper gridding and achieving order in the geometry and appearance of the element, we solve the problem. To reach the desired number of elements, we consider a main parameter that was the goal of the analysis (for example, the maximum stress, or the displacement at the middle point), which is considered here as stress. In the next step, we increase the number of elements (the dimensions of each element are reduced) and repeat the solution again to measure the effect of this element reduction on the mentioned parameter. We continue the work until a compromise is established between time and the number of elements, in other words, by increasing the number of elements, there is no specific change in the answers.

Here, in terminology, we say that the solutions have converged and there is no need to use more elements, and increasing the number of elements does not help to increase the accuracy of the solution and only increases the solution time. Mesh refinement does not always lead to convergence. There are also problems where the process of convergence of solutions in Abaqus changes to a sinusoidal shape with the increase of the number of elements, but the above technique is useful in many problems. Which is elemented in the final model, the fixture has 34096 elements, screws 24923 elements, bone 143287 elements and abutment 5200 elements. In order to find the appropriate number of elements, the modeled structure was analyzed in several stages with the number and density of different elements.

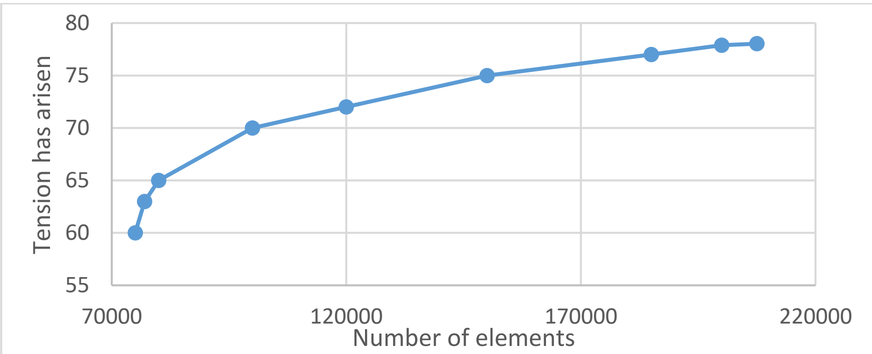


Diagram 1) Mesh convergence diagram

In table number 1, we define how to choose the type and size of elements, define mechanical properties and how to apply force. [10, 11, 14]:

Table 1) Mechanical properties of simulation components

Cortical bone		
	Modulus of elasticity	13700 MPa
	Poisson's ratio	0/3
	Density	1/74e-9

Titanium grade 4	
Modulus of elasticity	110000 MPa
Poisson's ratio	0/34
Density	4/5e-9
Titanium grade 4	
Modulus of elasticity	120000 MPa
Poisson's ratio	0/31
Density	4/5e-9
friction coefficient	0/2
Gradual applied force	100N
The size of the elements	0/5

Result

1. Von Misses stress distributed in the direct model of 0 mm

By applying the mentioned force, it was observed that the minimum stress in the inferior right part of the fixture was 0.3 MPa and the maximum stress in the anterior inferior part of the fixture was 42.6 MPa.

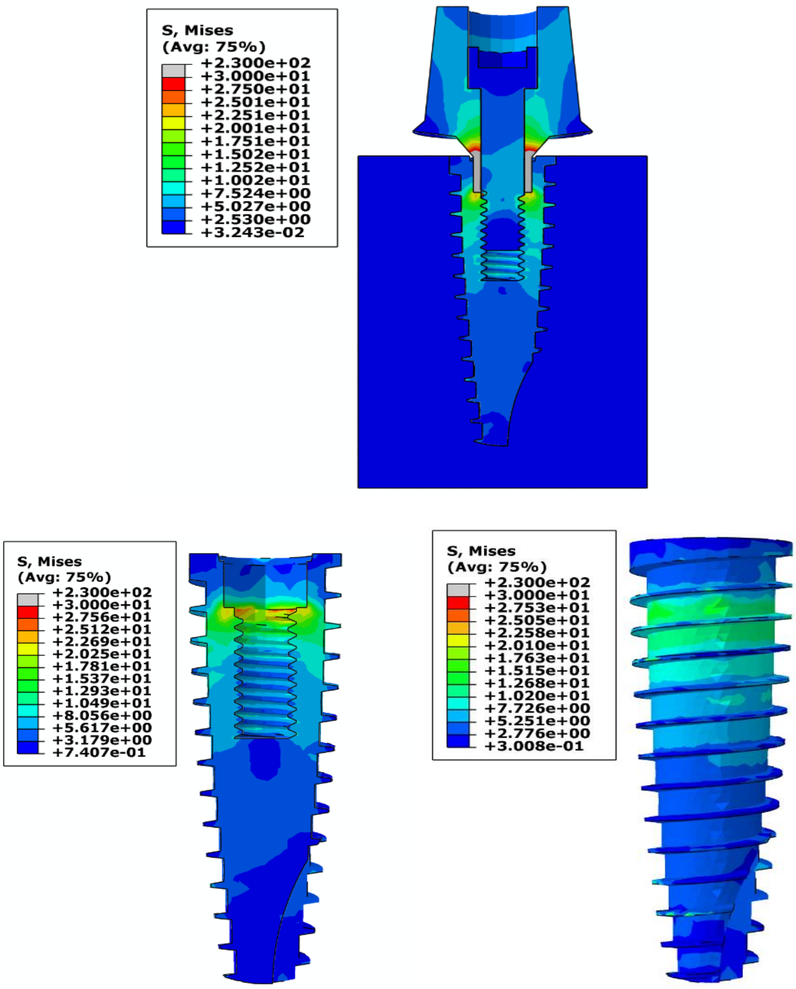


Figure 12) Checking the tension in the abutment with an angle of 0 degrees and a neck length of 0 mm

2. Von Misses stress distributed in the direct model of 1 mm

By applying the mentioned force, it was observed that the minimum stress in the inferior right part of the fixture was 0.2 MPa and the maximum stress in the anterior superior left part of the fixture was 40.4 MPa.

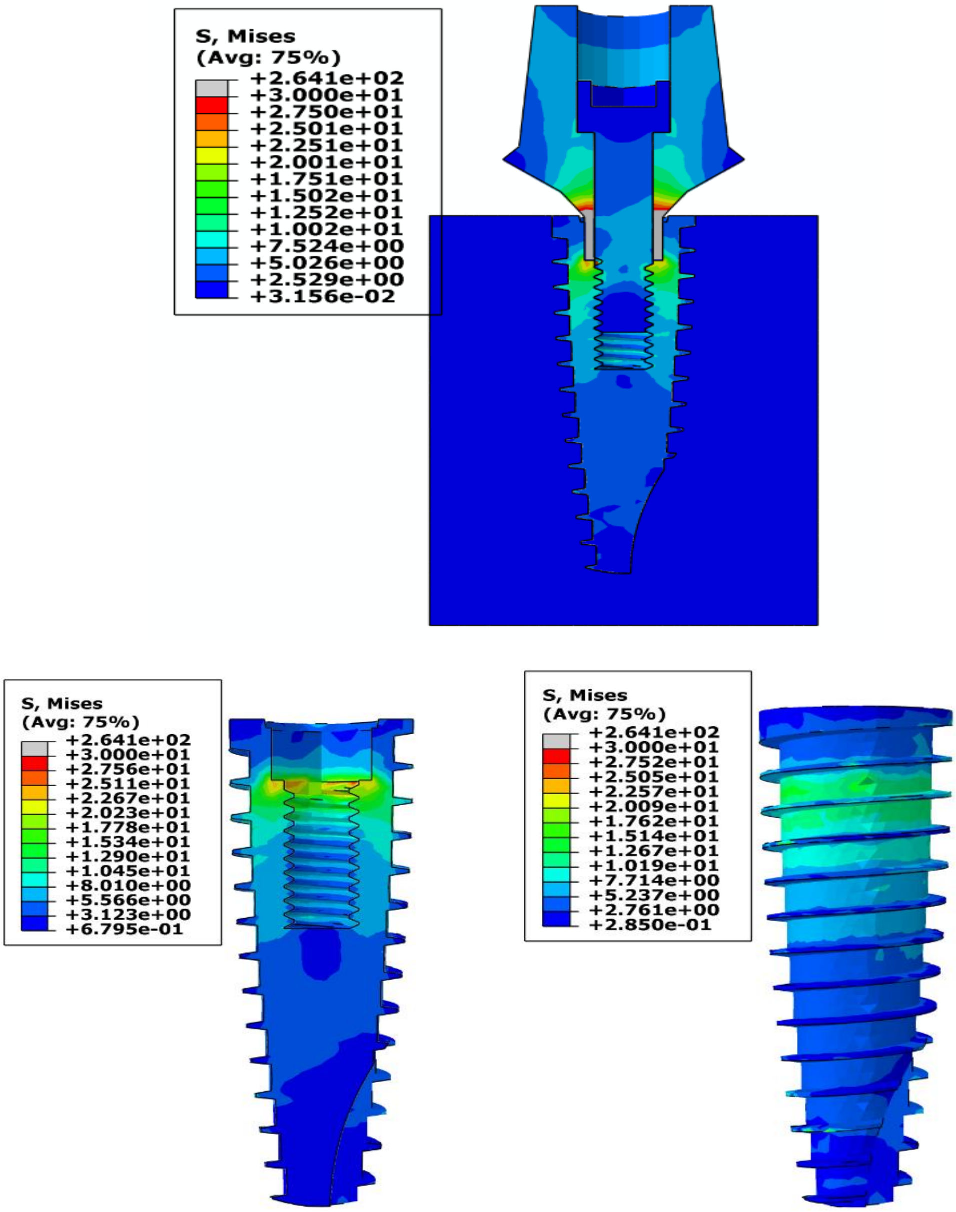


Figure 13) Checking the tension in the abutment with an angle of 0 degrees and a neck length of 1 mm

3. Von Misses stress distributed in the direct model of 2 mm

By applying the mentioned force, it was observed that the minimum stress in the inferior right (lower right) fixture was 0.3 MPa and the maximum stress in the upper loft (upper left) edge of the first outer thread of the fixture was 37.05 MPa.

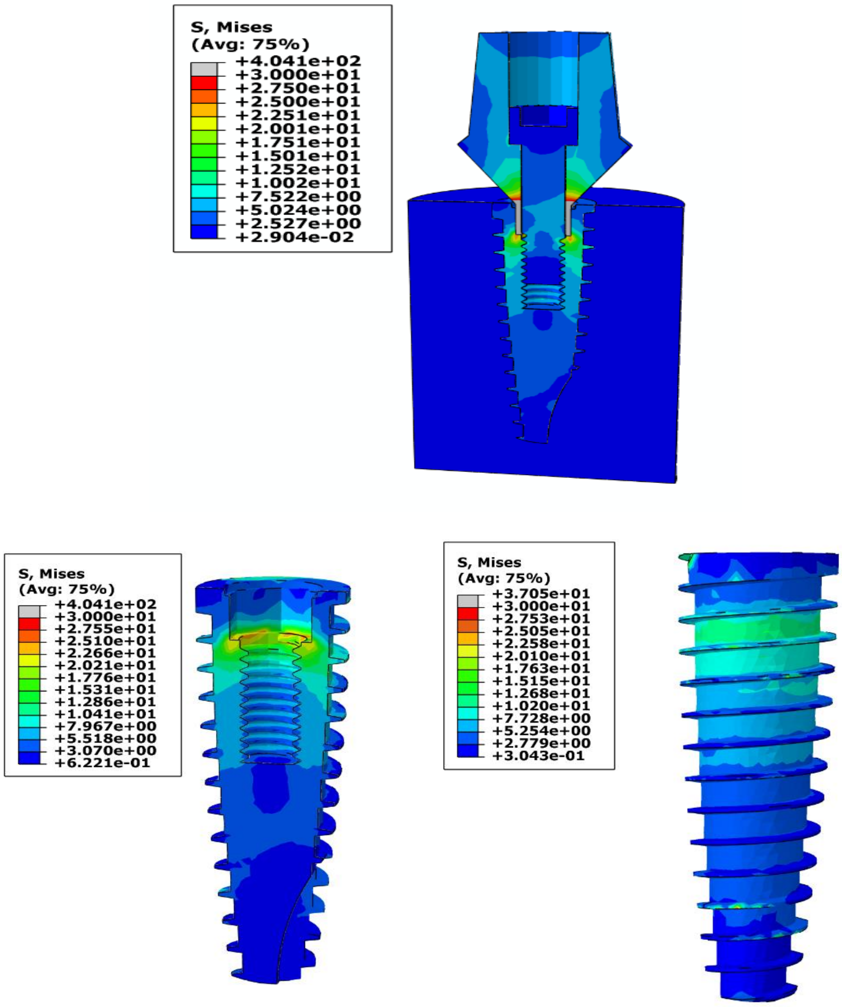
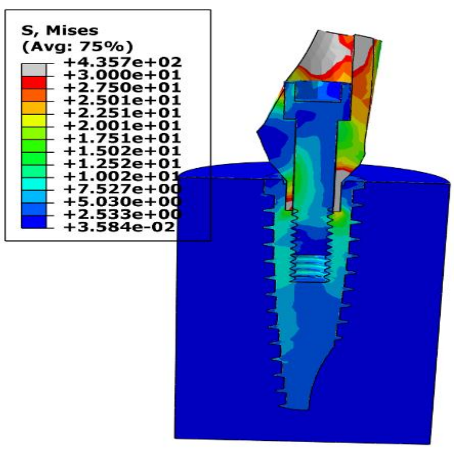


Figure 14) Checking the tension in the abutment with an angle of 0 degrees and a neck length of 2 mm

4. Von Misses stress distributed in the 15 degree 0 mm model

By applying the said force, it was observed that the minimum tension in the inferior right (lower right) fixture was 0.59 MPa and the maximum tension in the upper loft (upper left) part of the internal thread of the fixture was 70.15 MPa.



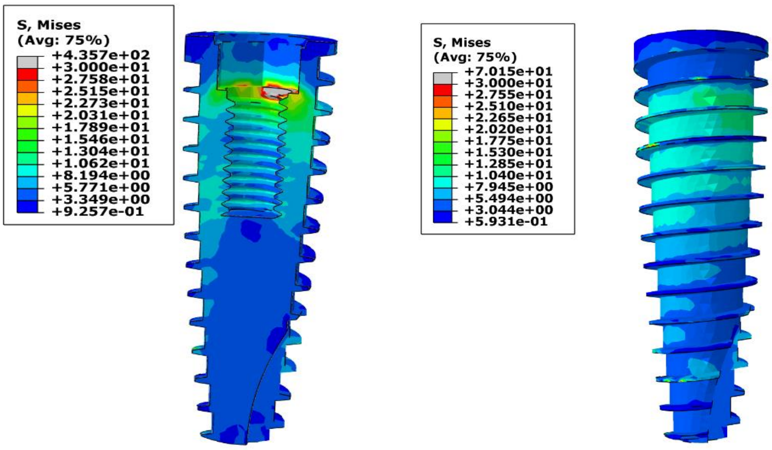


Figure 15) Checking the tension in the abutment with an angle of 15 degrees and a neck length of 0 mm

5. Von Misses stress distributed in the 15 degree 1 mm model

By applying the mentioned force, it was observed that the minimum tension in the inferior right (bottom right) of the fixture was 0.53 MPa and the maximum tension in the upper loft (upper left) part of the internal thread of the fixture was 68.13 MPa.

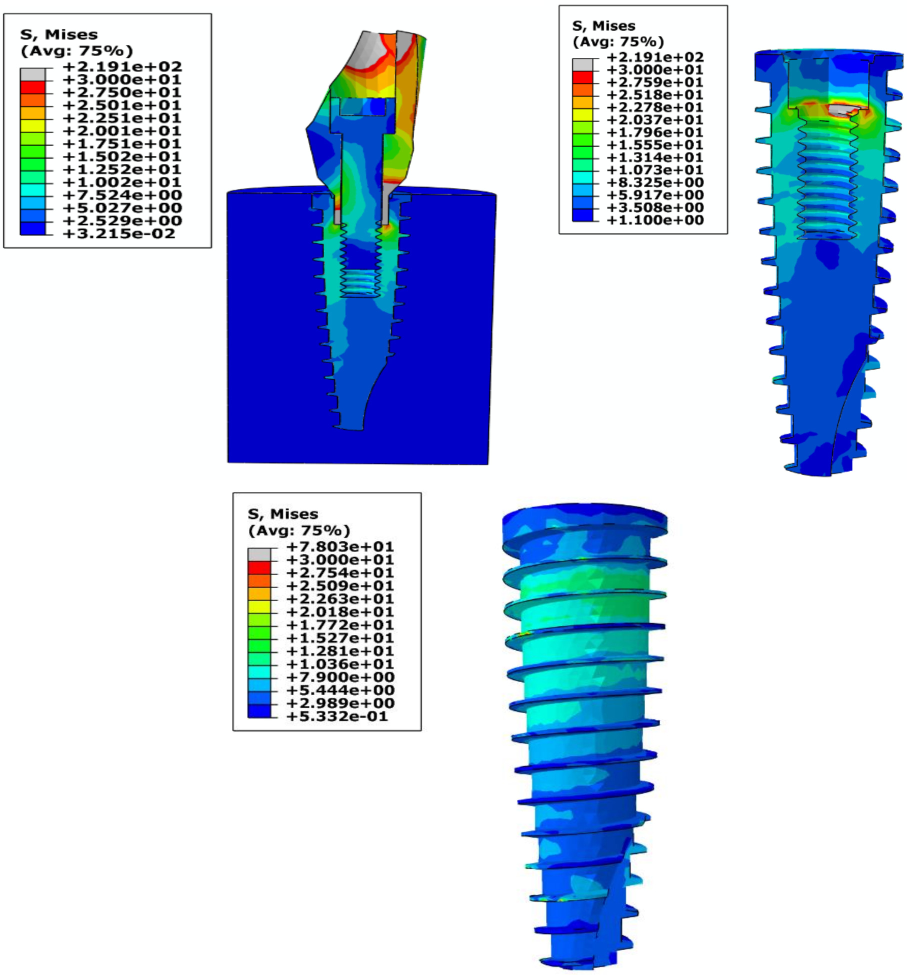


Figure 16) Checking the tension in the abutment with an angle of 15 degrees and a neck length of 1 mm

6. Von Misses stress distributed in the 15 degree 2 mm model

By applying the mentioned force, it was observed that the minimum tension in the inferior right (bottom right) of the fixture was 0.43 MPa and the maximum tension in the upper loft (upper left) part of the internal thread of the fixture was 68.1 MPa.

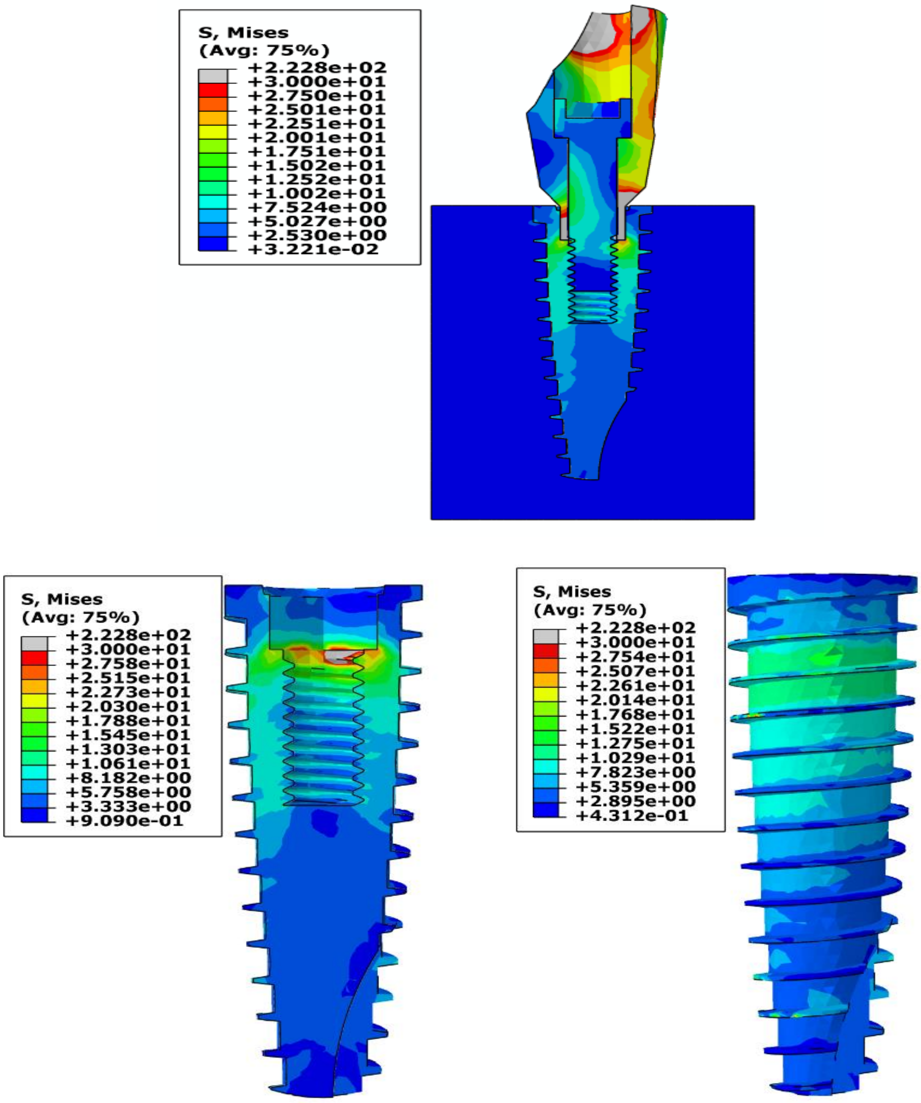


Figure 17) Checking the tension in the abutment with an angle of 15 degrees and a neck length of 2 mm

7. Von Misses stress distributed in the 25 degree 0 mm model

By applying the mentioned force, it was observed that the minimum stress in the inferior right (bottom right) of the fixture was 0.35 MPa and the maximum stress in the upper loft (upper left) part of the internal thread of the fixture was 71.07 MPa.

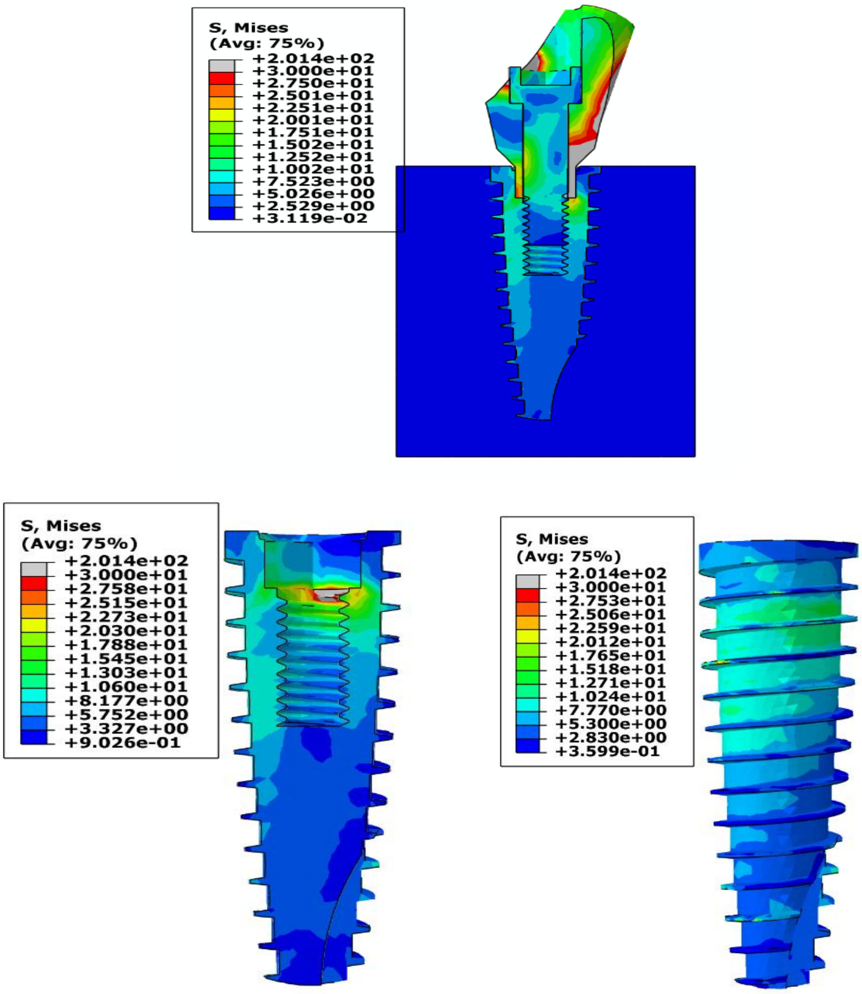
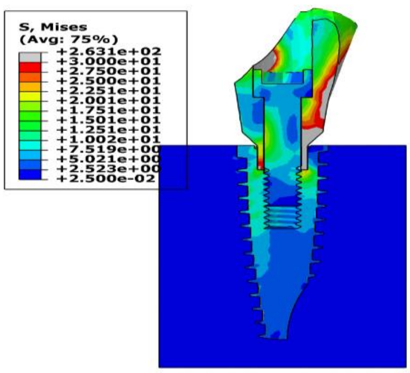


Figure 18) Checking the tension in the abutment with an angle of 25 degrees and a neck length of 0 mm

8. Von Misses stress distributed in 25 degrees 1 mm model

By applying the mentioned force, it was observed that the minimum tension in the inferior right (bottom right) of the fixture was 0.43 MPa and the maximum tension in the upper loft (upper left) part of the internal thread of the fixture was 68.31 MPa.



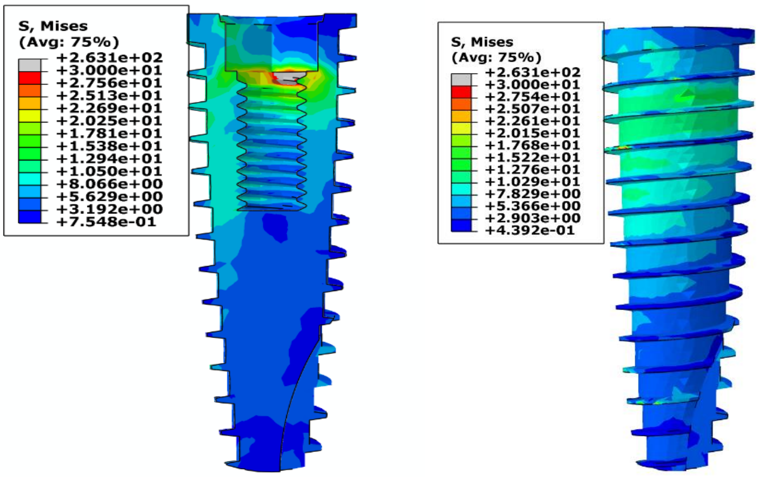


Figure 19) Checking the tension in the abutment with an angle of 25 degrees and a neck length of 1 mm

9. Von Misses stress distributed in the 25 degree 2 mm model

By applying the mentioned force, it was observed that the minimum tension in the inferior right (bottom right) of the fixture was 0.37 MPa and the maximum tension in the upper loft (upper left) part of the internal thread of the fixture was 68.25 MPa.

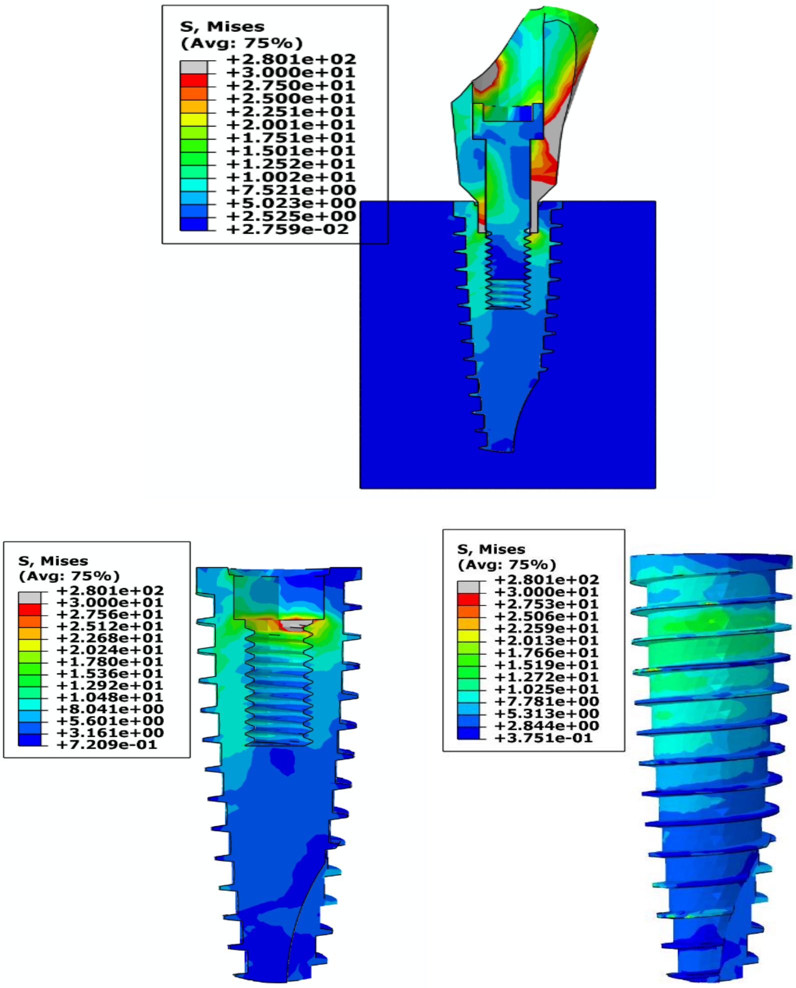


Figure 20) Checking the tension in the abutment with an angle of 25 degrees and a neck length of 2 mm

Discuss

Table number 2 of the maximum stress (MPa) generated in the fixture for all 9 abutment models designed in this research shows that as the length of the abutment neck increased, the stress decreased, but as the abutment angle increased, the stress also increased.

Table 2) the maximum stress generated in the fixture based on the abutment used

25 degrees	15 degrees	0 degrees	Abutment angle / Abutment neck length
71.07	70.15	42.6	0 Mm
68.31	68.13	40.4	1 Mm
68.25	68.1	37.05	2 Mm

Kavo et al. also reached similar results, so that according to Kavo et al.'s research, most researchers agreed that increasing the abutment angle leads to an increase in the stress on the fixture [15].

Ideally, implants should be placed parallel to each other, parallel to adjacent teeth and at the same height. However, in some cases, the anatomy and morphology of the remaining ridge prevents this from being achieved, and the implants must be placed at an angle [10], but note that the higher the angle, the higher the tension in the fixture.

In this research, we also investigated the effect of the height of the abutment neck on the tension in the fixture, which decreased with the increase in the length of the abutment neck, that is, the increase in the height of the abutment [16].

Studies and researches have already been done on the crown of the implant and they came to the conclusion that by increasing the vertical height of the crown and the ratio of the length of the crown to the implant, the stress on the abutment screw as the weakest member of the implant set is reduced [11].

Another research was conducted in connection with the crown, which concluded that changing the material of the crown changes the stresses and deformation values in cortical and spongy bones [17].

Of course, according to research, the success rate of implant implantation is about 90% [5, 14, 17-19]. However, in some people, this amount is greatly reduced and leads to the failure of the implant, which is mostly caused by infection, which is probably due to the specific diseases of the person being treated, which is related to the reduction of bone density and microbial adhesion to the surface of the implant [13].

Factors such as mechanical overload, abutment connection design, implant geometry, implant position, bone density have an effect on bone loss, which ultimately cause implant failure [8].

Conclusion

According to the analysis done in Abaqus software, it seems that as the abutment angle increases, the amount of stress increases, but as the length of the abutment neck increases, the amount of stress decreases. So it can be concluded that the tension has a direct relationship with the abutment angle and an inverse relationship with the length of the abutment neck. Therefore, it is recommended to dental implant specialists to use direct abutments with a neck length of 0, 1, and 2 mm depending on the person's jaw and mouth morphology, so that the amount of tension and possible damage is reduced to its minimum and as a result, the durability of dental implants increases.

Reference

[1] Robinson D, Aguilar L, Gatti A, Abduo J, Lee PV, Ackland D. Load response of the natural tooth and dental implant: A comparative biomechanics study. J Adv Prosthodont. 2019 Jun ;11 (3): 169 -178

[2] Aleksandrovich G A, Yur'evna D E, Taimushanovich K S, Igorevna R S, Vladimirovich E D. Use of foam form of 3% Ethoxysclerol in the venous

malformations treatment of the maxillofacial region . J Adv Pharm Educ Res. 2021;11(3):9- 14.

[3] Zakirulla M, Bakri HM, AlKhammash SMH, Alqahtani AHA, Alqahtani SJM, Alqahtani SSJ, et al. Knowledge of Dental Students Towards Implant Placement for Missing Permanent Teeth: A Cross-Sectional Study. Ann Dent Spec. 2022;10(4):56-60.

[4] K. Kniha, K.A. Schlegel, H. Kniha, A. Modabber, F. Hölzle, K. Kniha, Evaluation of peri-implant bone levels and soft tissue dimensions around zirconia implants-a three-year follow-up study . Journal: International Journal of Oral and Maxillofacial Surgery - Volume 47, Issue 4, April 2018, Pages 492-498

[5] Yuji Sato, Noboru Kitagawa, Akio Isobe, Implant treatment in ultra-aged society. Journal: Japanese Dental Science Review - Volume 54, Issue 2, May 2018, Pages 45-51

[6] V.E. de Souza Batista, A.J. Vechiato-Filho, J.F. Jr, M.V. Sonego, F.R. Verri, D.M. dos Santos, M.C. Goiato, E.P. Pellizzer, . Clinical viability of single implant-retained mandibular overdentures: a systematic review and meta-analysis. Journal: International Journal of Oral and Maxillofacial Surgery - Volume 47, Issue 9, September 2018, Pages 1166-1177

[7] Aifang Han, James K.H. Tsoi, Flávia Pires Rodrigues, Julian G. Leprince, William M. Palin, Bacterial adhesion mechanisms on dental implant surfaces and the influencing factors. Journal: International Journal of Adhesion and Adhesives - Volume 69, September 2016, Pages 58–71

[8] hriya Nimbalkar, Pankaj Dhatrak b, Chinmay Gherde a, Srujana Joshi. A review article on factors affecting bone loss in dental implants. Accepted 19 July 2020

[9] Alrakkad IA, Alrakkad RA, Altamimi MS, Alshammari NM, Alghuraymil AAS, John MAM, et al. Review on Dental Implant and Infection Management Approach. Arch Pharm Pract. 2022;13(1):37-9

[10] Mahmoudi. Mojtaba. The effect of abutment angle on stress distribution in dental implants. The second international conference on interdisciplinary research in electrical engineering, computer, mechanics and mechatronics in Iran and the Islamic world. September 30, 2018

[11] Hossein Netaj Miandeh Mojtabi, Nili Ahmadabadi Munira, Mahabadi Maitham. Dynamic study of the effect of increasing the vertical height of the crown on the stress created in the abutment screw and the bone around the short dental implant behind the mandible using finite element analysis method, number 2, period 43, winter 2018

[12] J.S. de Mello, F. Faot, G. Correa, O.L. Chagas Júnior, Success rate and complications associated with dental implants in the incisive canal region: a systematic review. Journal: International Journal of Oral and Maxillofacial Surgery - Volume 46, Issue 12, December 2017, Pages 1584-1591

[13] Wedad Qassadi1, Tahani AlShehri 1, Amani Alshehri 1, Kholud ALonazi 1, Ibtihal Aldhayan. Review on Dental Implant ology. The Egyptian Journal of Hospital Medicine (April 2018) Vol. 71(1), Page 2217-2225

[14] Tarek A. Soliman, Raafat A. Tamam, Salah A. Yousief, Mohamed I. El-Anwar, . Assessment of stress distribution around implant fixture with three different crown materials. Journal: Tanta Dental Journal - Volume 12, Issue 4, December 2015, Pages 249–258

[15] Kao Hc, Gung YW, Chung TF, Hsu ML. The influence of abutment angulation on micro motion level for immediately loaded dental implants: a 3-D finite element. 2008

[16] Syam S, Maheswari U. Prevalence of Incidental Findings in Maxillary Sinus Using Cone Beam Computed Tomography – A Retrospective Study. Pharmacophore. 2022;13(6):9-13.

[17] Carlos Nelson Elias, Daniel Jogaib Fernandes, Celso R.S. Resende, Jochen Roestel, Mechanical properties, surface morphology and stability of a modified commercially pure high strength titanium alloy for dental implants. Journal: Dental Materials - Volume 31, Issue 2, February 2015, Pages e1–e13

[18] Emil Nutu, Saad Ahmad, Stefan Pastrama. Influence of bone elastic properties on the predicted stress distribution in the dental implant vicinity. Materials Today: Proceedings 4 (2017) 5904–5908

[19] C.A.A. Lemos, F.R. Verri, R.S. Cruz, J.F. Santiago Júnior, L.P. Faverani, E.P. Pellizzer, . Survival of dental implants placed in HIV-positive patients: a systematic review. Journal: International Journal of Oral and Maxillofacial Surgery - Volume 47, Issue 10, October 2018, Pages 1336-1342.