Effect of angulated abutments on stress concentration at the implant abutment interface: a finite element analysis

Corresponding author

M. Oswal
Consultant Prosthodontist, Pune, India

Abstract

Background
In many clinical situations, there is a difference between the long axis of the implant and the long axis of the planned tooth to be replaced. Such clinical situations led to advent of angulated abutments. There have been Finite Element Analysis (FEA) studies for analyzing the stress distribution and the pattern of micromotion around the implants when angulated abutments were used, but not much literature is available with respect to stress accumulation at the implant abutment interface. Hence this study is being taken up to analyze the stress concentration at the implant abutment interface when different angulation of the abutment is used.

Aim
To study the stress concentration at the implant abutment interface with various angulated abutments.

Method and Material
Finite element Model assemblies were created to simulate single implant placed in mandibular posterior region. Different abutment angulation were tested keeping the length and diameter of the implant fixture constant. Force of 200 N was applied to simulate occlusal force. Von Mises stress were recorded foe each model at the implant abutment interface.

Results
As the abutment angulation increased, maximum von Mises stress decreased at the implant abutment interface.

Conclusion
The overall comparison showed as the implant fixture diameter is increased the von Mises stress at the implant abutment interface decreased, and suggested that appropriate diameter is essential whenever angled abutments are selected for rehabilitation.

Authors

M. Shetty[1]
C. Hegde[2]
N. John[3]

1
Professor and Head of the Department. Department of Oral Implantology. A B Shetty Memorial Institute of Dental Sciences, NITTE (Deemed to be University), Karnataka, India

2
Professor and Head of the Department. Department of Prosthodontics and Crown & Bridge. A B Shetty Memorial Institute of Dental Sciences, NITTE (Deemed to be University) , Karnataka, India

3
Lecturer. Department of Prosthodontics and Crown & Bridge. A B Shetty Memorial Institute of Dental Sciences, NITTE (Deemed to be University) , Karnataka, India

Keywords

Implant abutment interface, angled abutments, finite element analysis

DOI

10.23805/JO.2024.635

INTRODUCTION

The concept of osseointegration and introduction of dental implants has changed the course of treatment planning in rehabilitation of edentulous spaces. In case of removable dental prosthesis, adaptation to them might be difficult for certain individuals in which case implant supported rehabilitation would provide better stability as well as psychological benefit(1). In an ideal situation the implant fixture should be aligned parallel to the existing natural teeth and or dental implants so that the masticatory forces are directed axially(2). However there are certain clinical situations which interfere with the placement of dental implants in ideal positions in the jaws(3).
The morphology of the remaining alveolar bone dictates the direction of the implant fixture placement. Similarly, esthetics and function would determine the positioning of the teeth during replacement of the missing teeth. In most of the cases clinicians encounter deviation between the angulation of the end osseous implant and the angulation of the tooth being replaced.
This would arise due to inadequate width and height of the remaining alveolar bone or due to the presence of various anatomical structures like paranasal sinus in the maxillary jaw, the course of the mandibular nerve would prevent positioning of implants in the jaws(2). Management of scenarios like these would include augmentation of the residual ridge, sinus lift procedures, nerve repositioning, using zygomatic implants or using the various angulated abutments(4).
In recent times use of angled abutments has gained popularity due to its numerous advantages like facilitating parallelism between nonparallel dental implants, reducing the treatment time and cost by avoiding the augmentation procedures and helping the clinician to treat maximum patients by bypassing the anatomical landmarks(2,4,5). The consequence of angulation of abutment on stress distribution is a matter of debate. Various studies have concluded that angled abutments exert increased stress on the implant and surrounding bone(6,7,2).
However other studies (8,9,10) have shown that application of angled abutments results in favorable stress in the adjacent bone. Abutment angulation is an important biomechanical parameter playing a role in the long-term success of implants.
It is important to understand the behavior of angled abutments and their associated stress trajectories within the preimplant bone and along the dental implant Hence, the aim of this study was to measure and compare the stress at the implant-abutment interface with abutment angulations of 0˚,15˚ and 30˚ by the means of finite element analysis on implants placed in the posterior region of the mandibular jaw.

	Elements	Nodes
3.5 mm - Zero	429729	547682
3.5 mm -15 degrees	438180	548273
3.5 mm - 30 degrees	439412	552960
5 mm - Zero	460606	588893
5 mm -15 degrees	471055	596906
5 mm - 30 degrees	478287	603690

Tab. 1 Number of nodes and elements for each model

MATERIAL AND METHODS

A solid 3 – dimensional finite element model of mandibular arch was obtained using a computerized tomographic (CT) scan data of an edentate patient. The mandibular arch was modelled with outer cortical bone having a 2mm thickness and the inner bone volume represented the cancellous bone (D2 Bone type) Section of the bone at the molar region was extracted following which mesh models were obtained with the desired nodes and elements for the ease of interpretation during analysis (Table 1) (Fig. 1A-1B). Threaded root form implants made of

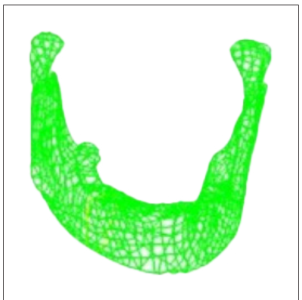


Fig. 1A Shows a geometrical model for the mandibular arch.

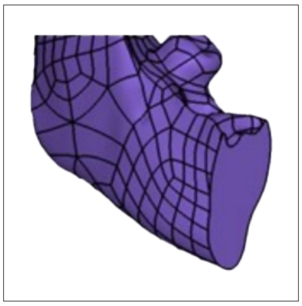


Fig. 1B Section made at the molar region.

Fig. 1C Three-dimensional view of Cortical and Cancellous bone at molar region.

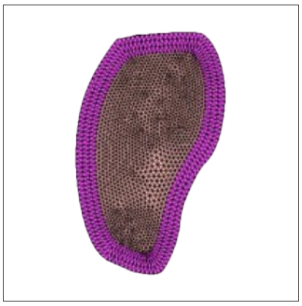


Fig. 1D Cortical and cancellous bone separation (2 mm thickness of cortical bone)

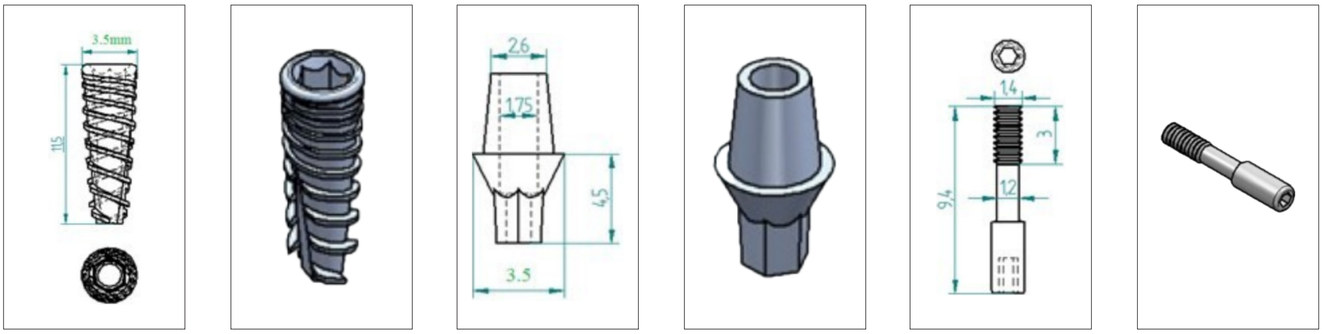


Fig. 2 Design of the implant and abutment for 3.5 mm diameter having a screw retained connection with the design of the screw connection.

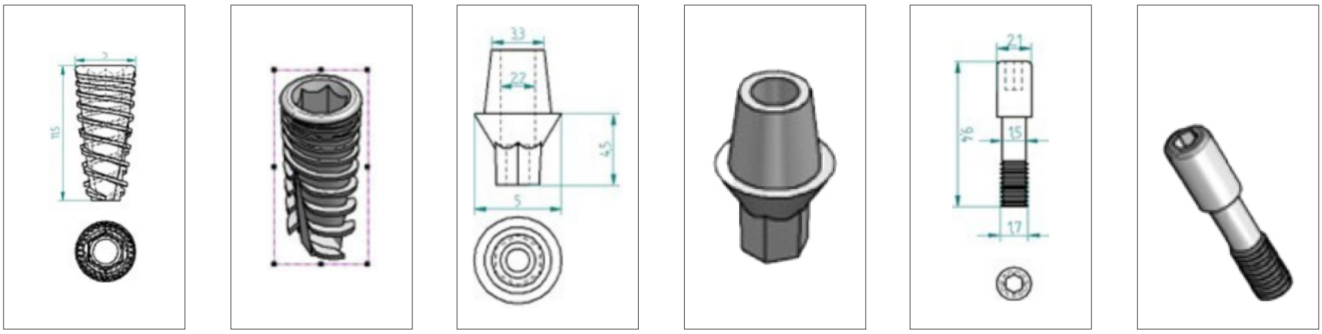


Fig. 3 Design of implant and abutment for 5mm diameter having a screw retained connection with the design of the screw connection.

Model no	Implant diameter	Implant length	Abutment angulation
Model 1	3.5mm	11.5mm	0 degree
Model 2	3.5mm	11.5mm	15 degrees
Model 3	3.5mm	11.5mm	30 degrees
Model 4	5mm	11.5mm	0 degree
Model 5	5mm	11.5mm	15 degrees
Model 6	5mm	11.5mm	30 degrees

Tab. 2 Configuration of various models used in the 3d Finite element analysis

titanium of dimensions 3.5 mm diameter and 11.5mm length and 5mm diameter and 11.5mm length were developed using the 3D modelling software (Solid Edge V19, Siemens AG, Munich, Germany). The implants had complete osseointegration at the bone implant interface. An abutment made of titanium having a screw connection with the implant was designed appropriately for the 3.5mm diameter and 5mm diameter implant(Fig 2, 3). The abutment angulations were 0-degree, 15 degree, and 30 degrees(Table 2).
The implant and abutment assembly along with a screw connection is implanted within the bone at the desired region (Fig 4, 5). The principal of a FEA analysis is to break the model into numerous finite nodes for evaluation. Hence, a finite element model having a 3D configuration corresponding to the

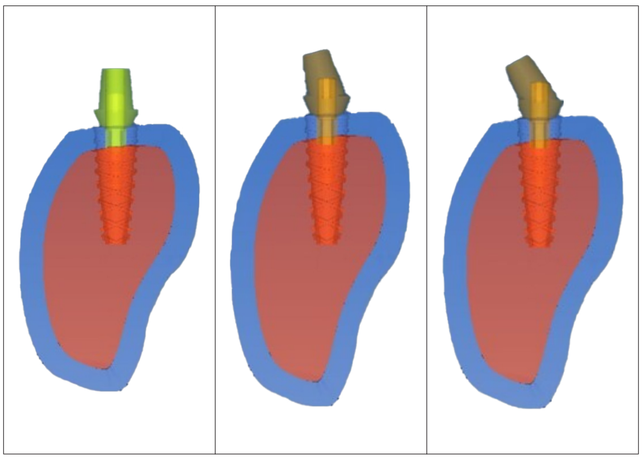


Fig. 4 Assemblies of model 1, 2, 3 - 3.5 mm diameter implant with 0-degree, 15-degree, 30-degree angled abutment.

geometric model was meshed using the HyperMesh 11 software (Altair Engineering Inc, Troy, MI, USA). Since the model is not symmetrical in geometry free meshing is used.
The materials used for model construction were considered to be homogenous, isotropic and linearly elastic to each other. These particularly included the outer cortical bone, inner cancellous bone, the end osseous implant along with the abutment. The analysis was performed on two material properties Young's Modulus and Poisson's ratio(Table 3). Boundary conditions were fixed. This is done particularly to prevent the objects motion upon which external load is applied. The bottom end support is eliminated to appreciate the bending of the model. This helps to achieve a closer clinical representation of the scenario. Abutment was subjected to a load of 200 N in the axial direction. This stress was within the physiologic limit. The software will present stress values at the desired area which will be tabulated and analyzed using ANSYS (version 18.1, Ansys, Canonsburg, PA, USA).

RESULTS

The overall stress generated in six different models when an axial force of 200 N is applied is represented the highest stress of 180.684 Mega Pascal (MPa) is seen with 3.5mm diameter -30-degree angled

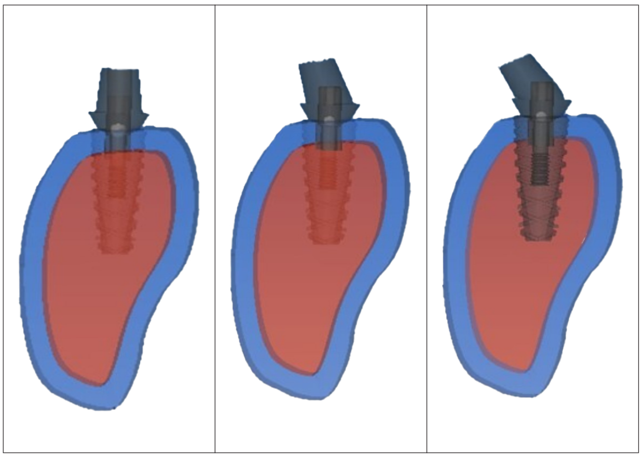


Fig. 5 Assemblies of model 4, 5, 6 - 5 mm diameter implant with 0-degree, 15-degree, 30-degree angled abutment.

Material	Youngs Modulus (GPa)	Poissons Ratio
Titanium	110	0.35
Cortical Bone	13.7	0.30
Cancellous Bone	1.37	0.30

Tab. 3 Material properties.

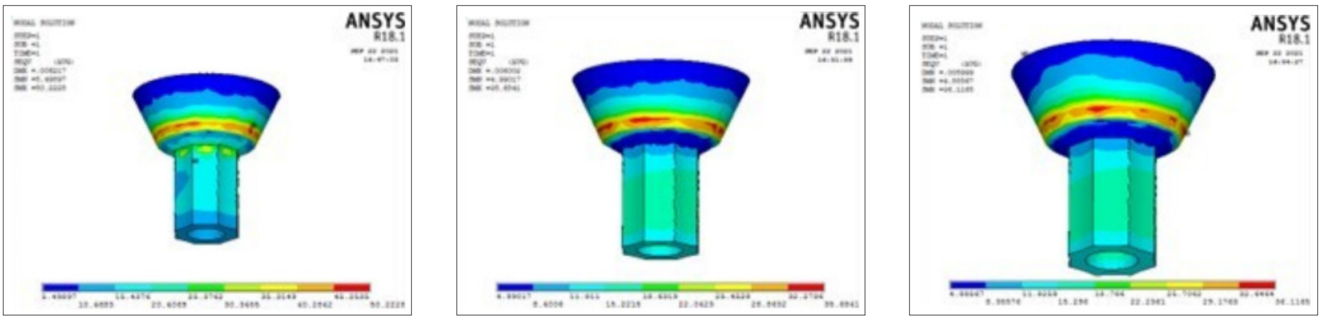


Fig. 6 Shows implant-abutment interface and Von Mises Stress for 3.5mm – 0˚, 3.5 mm – 15˚ and 3.5 mm – 30 ˚.

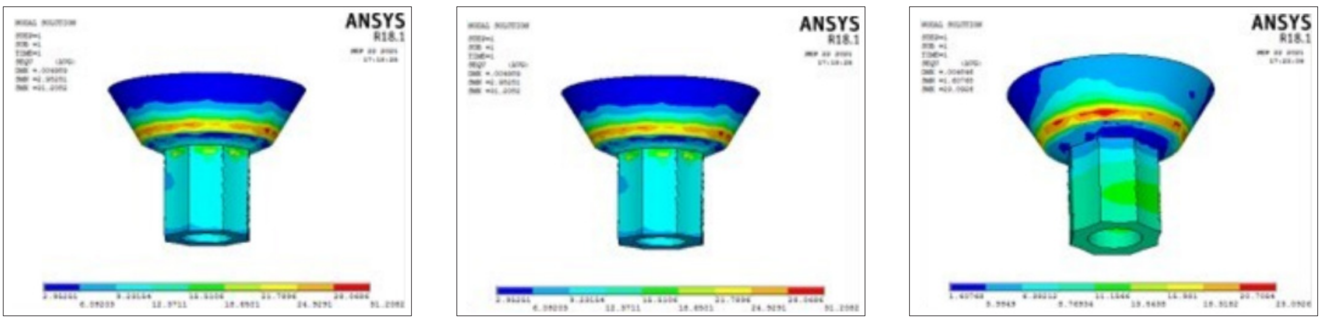
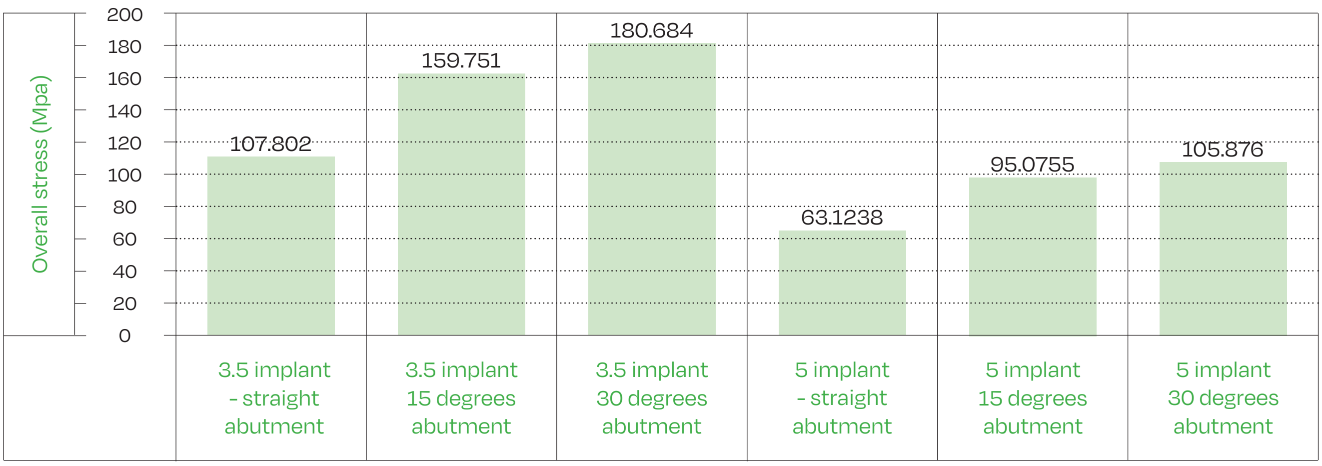
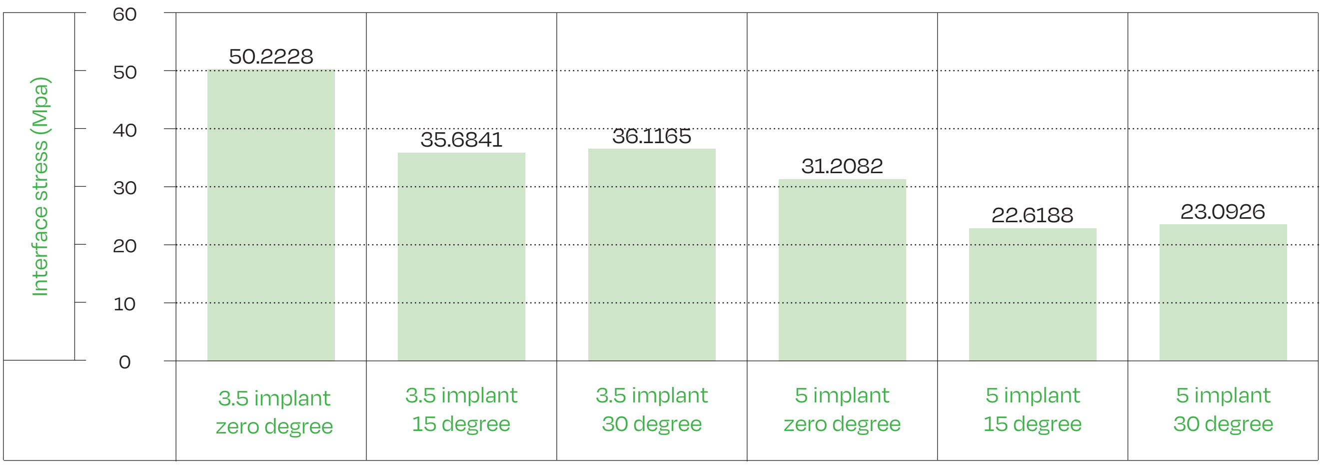


Fig. 7 Shows implant-abutment interface and Von Mises Stress for 5mm – 0˚, 5 mm – 15˚ and 5 mm – 30 ˚.



Graph. 1 Representation of the overall stress generated in six models when an axial force of 200 N is applied.



Graph. 2 Comparison of implant-abutment Von Mises Stress for 3.5 mm and 5mm implant diameters.

abutment. The 5mm -straight abutment assembly shows the least amount of overall stress of 63.1238 MPa. (Graph 1). With an increase in the abutment angulation the overall stress values increased in both the 3mm diameter group and the 5 mm diameter group.
Furthermore, the vales of overall stress suggest that as the diameter increased the overall stress values decreased.
The results of the Von Mises stress at the implant-abutment interface are present in (Graph 2). The highest value of Von Mises stress is recorded with 3.5mm diameter implant with zero-degree abutment angulation. Least amount of interface stress was observed with 5mm diameter implant with abutment having 30-degree angulation.
Comparison of the results within the 3mm diameter implant models suggest as the angulation was increased from 0, 15 and 30 degree the interface stress decreased. Similar observation was seen with the 5mm diameter implant model.

DISCUSSION

Endosseous implants are one of the most widely used treatment modality for replacement of the missing natural teeth. Understanding the biomechanics related to these structures is essential because the endosseous Implants are exposed to numerous loading conditions within the oral cavity during various oral functions [11]. Biomechanics is an important component influencing the survival of these Endosseous Implant and has a direct influence on the load transferring capabilities of the Endosseous Implant to the surrounding bone. [12,13] Factors which would influence the biomechanics are the length and diameter of the endosseous implant, the surface morphology and the angulation of the abutment used. All the above-mentioned parameters can be controlled by the clinician. Therefore, using

these parameters with utmost caution is desired to prevent long term mechanical failures.
Finite element analysis (FEA) is considered a valuable tool which gives the researcher an in-depth understanding of the stress distribution sequence in and around the dental implant. The researcher can evaluate the stress and strain levels quantitatively in any area on the model under analysis(14). FEA can be done with a 2 or 3 D model, but Meijer et al. (15) proposed particularly the use of 3D model with the reason being in a 2D model the object is symmetrical whereas the Endosseous implants have an asymmetrical geometry. Therefore, in the present study 3D models were developed. When a particular load is applied the stress at the implant-abutment interface comprise of Von Mises stress and strains which are compressive or tensile in nature. On Mises stress are a consolidation of normal along with shear stresses which help clinician presume the yield stage of a material under compound load conditions(16). Therefore, stress accumulation at the implant-abutment interface may cause to prosthetic failure such as screw loosening, screw fracture and abutment fracture.
As the foundation of osseointegration began to amalgamate, researchers shifted their attention to implant prosthetic interface to address the problems of screw loosening, fracture of the screw, implant fracture or prosthesis fracture because the incidence of these prosthetic failures was more when compared to failures with reduced osseointegration (17). Improper load application onto the implant may result in abnormal stress distribution and may hamper the survival of implant. Extensive evidence is present in literature regarding the use of angled implant-abutment and their associated surge in the stress values in the peri implant bone(6,7,18,19).
All of these studies conclude that there is increased stress in the bone adjacent to the implant but this stress being within the physiologic limit.
The finding of a few studies has raised a debate about angled abutments. In a 2D FEA study by Nothdurft et al.(8) which compared the bone strain for straight and angled abutment, reported a 15 % higher maximum bone strain for straight abutment. Similar positive findings for angled abutments have been reported by Eger et al and Sethi et al.(4,20).
Hence this study was undertaken to evaluate the effect of angled abutment on stress concentration at the implant-abutment interface by the means of FEA. The study consisted of two different diameter implants 3.5 mm and 5mm with the length being the same for both 11.5 mm. The abutment angulation under were 0-degree, 15 degree, and 30 degrees respectively. The 6 different model assemblies were developed as seen in (Fig. 4, 5).
Findings in literature suggest that force of mastication in the posterior region of a natural tooth in 220N (21). Therefore assemblies were then subjected to 200 N of static axial loading. The FEA (ANSYS) software gave different quantitative values at all the locations. Von Mises stress was assessed at the implant-abutment interface.
According to the results of this study the maximum Von Mises stress at the interface was seen with 3.5mm -0-degree abutment and least stress was seen with 3.5 mm -15-degree angled abutment. When the abutment was straight (0 ˚) Von Mises stress at interface was 50.222 MPa, it decreased to 35.6841 MPa when the angulation increased to 15˚and then Increased to 36.1165 MPa when the angulation increased 30˚.As the angulation progressed from 0˚ to 15˚ the Von Mises stress decline and as the angulation changed from 15˚ to 30˚ the Von Mises stress increased at the interface (Graph 2).
Furthermore, the Von Mises stress at interface for 5mm 0˚ assembly had highest Von Mises stress value when compared to 15 ˚and 30 ˚ abutment. The 5mm 0˚assembly showed 31.2082 MPa of Von Mises strain and as the angulation increased from 0˚ to 15 ˚ the Von Mises stress decreased. 23.0926 MPa of interface stress was recorded with the 5mm 30 ˚ abutment assembly. The finding is similar to 3.5 mm diameter assemblies in which as the angulation changed from 0˚ to 15˚ the Von Mises stress decreased and as the angulation changed from 15˚ to 30˚ the Von Mises stress increased at the interface. One of the reasons for this finding could be that angled abutment had a considerable influence on the pattern of stress distribution. The FEA analysis of 0 ˚ abutment angulation with both 3.5 mm and 5 mm shows a more symmetrical pattern of stress distribution. Whereas if we compare with 15 ˚ and 30 ˚ abutment angulation of both 3.5 mm and 5mm diameter assemblies they show asymmetrical type of Von Mises stress distribution. Therefore, this suggests that the angled abutment used to correct the direction of implant placed in clinical situations would result in better distribution of stress and strain.
Literature search presented no study assessing the effect of abutment angulation on implant-abutment interface.
Comparison can be made with other studies which assess the effect of angled abutment with respect to peri implant bone stress.
Assessment of effect of angled abutment on the preimplant bone has been studied by Cardelli P et al. (19) and the author concluded that these angled abutments if indicated should not be more that 25 degree and if achievable should be used in anterior region where it is presumed that forces over the endosseous implant are lower when compared with posterior region. This can be corelated to the finding of our study in which by using a wider diameter

implant the stresses at the abutment interface reduce and seems to be a valuable clinical information.
Bahuguna et al (18) systematically measured and compared the stress in alveolar bone around dental implants for different angled abutments when an axial as well as oblique load was applied with the help of FEA. The results of this study presented that as the angulation changes from 0 to 20 degree the amount of compressive as well as tensile stress increase, but these stresses are well within the physiologic limit of tolerance of alveolar bone.
The study by Tian et al (10) which aimed to access if angled abutments resulted in increased or decreased stress on the surrounding bone of a single dental implant by means of FEA. The author developed 4 simplified assemblies which reproduce the clinical situation. The implants were positioned in ideal axial orientation or at an angle. They concluded that angled abutments may result in decreased Von Mises stress. Saab et al (22) studied the effect of angled abutment on bone strain surrounding an implant in the anterior maxillary region by the means of FEA. The conclusion drawn from this study was that straight abutment resulted in 15 %higher maximum bone strain when compared to the angled abutment. The findings of the present study also concur with study of Saab et al, in which a higher abutment interface stress was observed with 3.5 mm and 5mm diameter with straight abutment assemblies.
Furthermore, based on the results of our study a comparison was made between the 3.5 mm diameter group and the 5mm diameter (Graph 2).
The highest Von Mises stress 50.228 MPa was recorded with 3.5mm-0 ˚angled, 35.6841 MPa with 3.5mm-15˚ and 36.1165 MPa with 3.5 mm -30˚group. With the 5mm diameter group 31.2082 MPa stress with 0 ˚, 22.6188MPa with 15 ˚ and 23.0926 MPa with 30˚was recorded. An interesting observation was made here with respect to the diameter of the implant. As the diameter of the implant increased from 3.5mm to 5mm the Von Mises stress at the interface decreased.
The stress value dropped from 50.228 MPa to 31.2082 MPa as the diameter of the implant increased. The least Von Mises stress was recorded with the 5mm -15˚ abutment assembly. From the above findings we can take into consideration that the diameter of the abutment has a role to play in the stress distribution. Shetty et al (23) in their review on implant design and stress distribution have stated that implant diameter is a more critical factor when compared to its length. Misch et al (24) reported that increasing the diameter of the implant by 1 mm would lead to an increase in the total surface area by 35 % keeping the length as constant. This in turn results in effective stress distribution to the surrounding bone. The wider diameter implants also have an increased contact with the surrounding bone and provide resistance to stresses. However, there are certain limitations of the present study. The study is a Finite element analysis and all the structures were homogenous and isotropic. The implant was assumed to have 100 % osseointegration which would differ from the clinical situation. Only an Axial load was applied, which differs from the dynamic loading occurring in the oral cavity oral environment. No prosthesis was placed over the abutment which might also influence the stress distribution.
Further research needs to be carried out which compares the interface stress when dynamic loading is applied over the assemblies. This would help to make the results of this more valid. Modeling the bone as an anisotropic and nonhomogeneous regenerative tissue that responds to stress by resorption or regeneration under load would also be an improvement in current finite element models to address the issues found in this study.

CONCLUSION

Within the limitations of the present study, it was concluded that:

1) The use of angled abutment resulted in decreased implant-abutment interface stress when compared to the straight abutment.
2) As the diameter of the implant increased from 3.5 mm to 5mm the Von Mises stress decreased.
3) The cortical bone Von Mises stress was higher with angled abutment when compared to straight abutment.

REFERENCES

1. Lundgren D, Laurell L. Occlusal force pattern during chewing and biting in dentitions restored with fixed bridges of cross-arch extension Vol. 13, Journal of Oral Rehabilitation. 1986. p. 191–203.
2. Cavallaro J Jr, Greenstein G. Angled implant-abutments: a practical application of available knowledge. J Am Dent Assoc. 2011 Feb;142(2):150–8.
3. Clelland NL, Gilat A. The effect of abutment angulation on stress transfer for an implant. J Prosthodont. 1992 Sep;1(1):24–8.
4. Eger DE, Gunsolley JC, Feldman S. Comparison of angled and standard abutments and their effect on clinical outcomes: a preliminary report. Int J Oral Maxillofac Implants. 2000 Nov;15(6):819–23.
5. Sethi A, Kaus T, Sochor P. The use of angulated abutments in implant dentistry: five-year clinical results of an ongoing prospective study. Int J Oral Maxillofac Implants. 2000 Nov;15(6):801–10.
6. Kao H-C, Gung Y-W, Chung T-F, Hsu M-L. The influence of abutment angulation on micromotion level for immediately loaded dental implants: a 3-D finite element analysis. Int J Oral Maxillofac Implants. 2008 Jul;23(4):623–30.
7. Clelland NL, Lee JK, Bimbenet OC, Brantley WA. A three-dimensional finite element stress analysis of angled abutments for an implant placed in the anterior maxilla. J Prosthodont. 1995 Jun;4(2):95–100.
8. Nothdurft FP, Doppler KE, Erdelt KJ, Knauber AW, Pospiech PR. Fracture behavior of straight or angulated zirconia implant-abutments supporting anterior single crowns. Vol. 15, Clinical Oral Investigations. 2011. p. 157–63.
9. Geng J, Yan W, Xu W. Application of the Finite Element Method in Implant Dentistry.

Springer Science & Business Media; 2008. 137 p.
10. Tian K, Chen J, Han L, Yang J, Huang W, Wu D. Angled abutments result in increased or decreased stress on surrounding bone of single-unit dental implants: a finite element analysis. Med Eng Phys. 2012 Dec;34(10):1526–31.
11. Nokar S, Jalali H, Nozari F, Arshad M. Finite Element Analysis of Stress in Bone and Abutment-Implant Interface under Static and Cyclic Loadings. Front Dent. 2020 Sep;17(21):1–8.
12. Carter DR, Van Der Meulen MC, Beaupré GS. Mechanical factors in bone growth and development. Bone. 1996 Jan;18(1 Suppl):5S – 10S.
13. Eazhil R, Swaminathan SV, Gunaseelan M, Kannan GV, Alagesan C. Impact of implant diameter and length on stress distribution in osseointegrated implants: A 3D FEA study. J Int Soc Prev Community Dent. 2016 Nov;6(6):590–6.
14. Bozkaya D, Muftu S, Muftu A. Evaluation of load transfer characteristics of five different implants in compact bone at different load levels by finite elements analysis. J Prosthet Dent. 2004 Dec;92(6):523–30.
15. Meijer HJ, Kuiper JH, Starmans FJ, Bosman F. Stress distribution around dental implants: influence of superstructure, length of implants, and height of mandible. J Prosthet Dent. 1992 Jul;68(1):96–102.
16. Finite element analysis: A Boon To Dental Research Vol. 6, The Internet Journal of Dental Science. 2009.
17. Segundo RMH, Oshima HMS, da Silva INL, Burnett LH Jr, Mota EG, Silva LL. Stress distribution of an internal connection implant prostheses set: a 3D finite element analysis. Stomatologija. 2009;11(2):55–9.
18. Bahuguna R, Anand B, Kumar D, Aeran H, Anand V, Gulati M. Evaluation of stress patterns in bone around dental implant for different abutment angulations under axial and oblique loading: A finite element analysis. Natl J Maxillofac Surg. 2013 Jan;4(1):46–51.
19. Cardelli P, Montani M, Gallio M, Biancolini M, Brutti C, Barlattani A. Angulated abutments and perimplants stress: F.E.M. analysis. Oral Implantol . 2009 Jan;2(1):3–10.
20. Sethi A, Kaus T, Sochor P, Axmann-Krcmar D, Chanavaz M. Evolution of the concept of angulated abutments in implant dentistry: 14-year clinical data. Implant Dent. 2002;11(1):41–51.
21. Sertgöz A, Güvener S. Finite element analysis of the effect of cantilever and implant length on stress distribution in an implant-supported fixed prosthesis. Vol. 76, The Journal of Prosthetic Dentistry. 1996. p. 165–9.:
22. Saab XE, Griggs JA, Powers JM, Engelmeier RL. Effect of abutment angulation on the strain on the bone around an implant in the anterior maxilla: a finite element study. J Prosthet Dent. 2007 Feb;97(2):85–92.
23. Shetty P, Yadav P, Tahir M, Saini V. Implant Design and Stress Distribution. Vol. 7, International Journal of Oral Implantology & Clinical Research. 2016. p. 34–9.
24. Misch CE. Implant design considerations for the posterior regions of the mouth. Implant Dent. 1999;8(4):376–86.