

# COMPARATIVE MEASUREMENT OF FRACTURE RESISTANCE OF VARIOUS KIND INTERNAL CONNECTION SYSTEMS

Topic: Material research

## Abstract

### Background:

Dental implants are a well accepted treatment for partially or totally edentulous subjects. Innovations through research have led to advancements in surgical and restorative techniques, improved surface features and restorative components. Dental implants typically use either internal or external connections with the crown. Although both connections are extensively used clinically, distinctly different stress distributions are produced within the crown. Although both connections are extensively used clinically, distinctly different stress distributions are produced within the crown. Clinicians have reported implant components linked to mechanical failure of crown and implant.

### Aim:

The purpose of this study was to compare the stress distribution characteristics of different abutment connections under occlusal loading, using 3-dimensional (FEM) Finite Element Method.

### Methods:

In this study three different implant brands (Dental Implant KA® and other four implants) were investigated. The strain distribution of different implant-abutment connection systems having same material properties were evaluated under same loading conditions, the advantages and disadvantages of each system were assessed. The investigations were performed using FEM (Finite Element Analysis Software) methods in a software based system.

### Results:

No differences were found between different implant brands.

### Conclusions and clinical implications:

In this study different Dental Implant KA implant samples were applied by means of finite element computational method. In the conclusion not a significant effect was observed.

## Background and Aim

The prosthetic elements have an important effect on the degree of implant osseointegration of the success, especially in terms of esthetics, quality of life and patient satisfaction, depends on the prosthetic elements. In particular, the implant-abutment connection is of great importance when it comes to long-term stability and a successful prosthetic restoration. Dental implants are particularly sensitive to lateral forces. Therefore the design of an internal connection system that neutralizes and distributes all forces equally on the entire implant is crucial. The aim of this preliminary study was to make a qualitative comparison and evaluation, by means of 3D-finite element analysis, the stress distribution produced by simulated load under a internal connection implant systems.

## Methods and Materials

The 3-D FEA is considered an appropriate method for investigation of the stress throughout a 3-D structure, and therefore this method was selected for bone and implants stress evaluation in this study. The software SOLIDWORKS (Yensoft, Ataşehir, İstanbul) was used for preprocessing, finite element analysis, and ANSYS 14.0 for postprocessing in the study. In this study five different implant brands (Dental Implant KA and other four implants) were investigated. The strain distribution of different implant-abutment connection systems having same material properties were evaluated under same loading conditions, the advantages and disadvantages of each system were assessed. The investigations were performed using FEM (Finite Element Analysis Software) methods in a software based system.

## References

- Segundo H, Oshima H, Silva I. Stress distribution of an internal connection implant prostheses set: A 3D finite element analysis. Stomatologija, Baltic Dental and Maxillofacial Journal, 11:55-59, 2009
- Staden CR, Loo YC, Johnson N. Comparative analysis of two implant-crown connection systems- A Finite Element Study. Applied Osseointegration Research-Volume 6, 2008.

Forces of 300 N, 200 N, 100 N, 50 N and 30 N were separately applied axially (AX), to the center of the internal connection totally 5 different FE model and the von Mises stresses (equivalent [EQV] stresses) in the structure were calculated.

## Results

The Course of Force	Vertical	Horizontal	Oblique
The Amount of Force (Newton)	100	30	50
Maximum Value of Stress (Von Mises N/m <sup>2</sup> )	1.068.586 .432.0	1.007.786.17 6.0	1.244.403.45 6.0

Table 1: The amount, direction and stress values of the forces used in the 3D Model

Regardless of abutment types and loading conditions, higher stress concentration was observed at the cortical bone. In cancellous bone, the highest stress was observed at apical portion and the maximum stress occurred at the implant neck. The higher internal stress was observed in the fixtures than in the bone. No differences were found between different implant brands. In this study different Dental Implant KA implant samples were applied by means of finite element computational method. In the conclusion not a significant effect was observed.

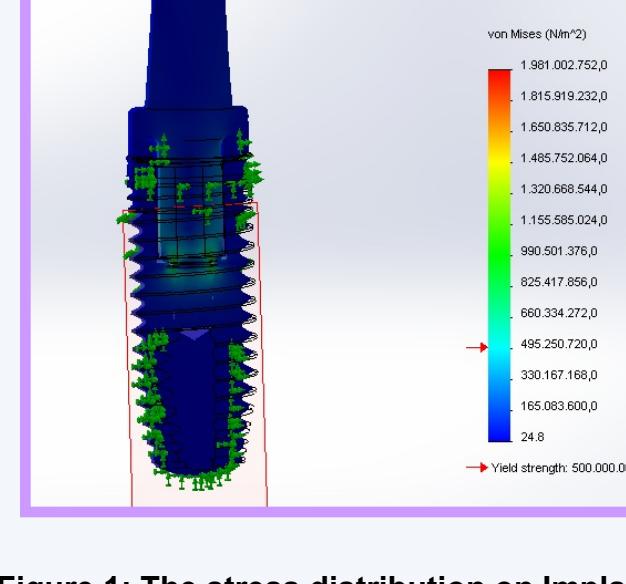


Figure 1: The stress distribution on Implant KA after applying angled 100 N force

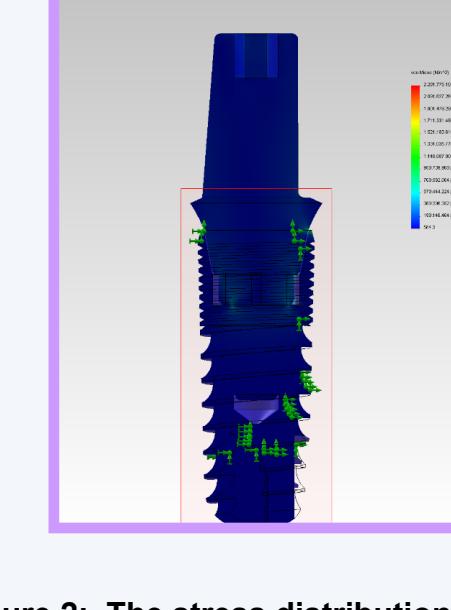


Figure 2: The stress distribution on Implant KA after applying angled 200 N force

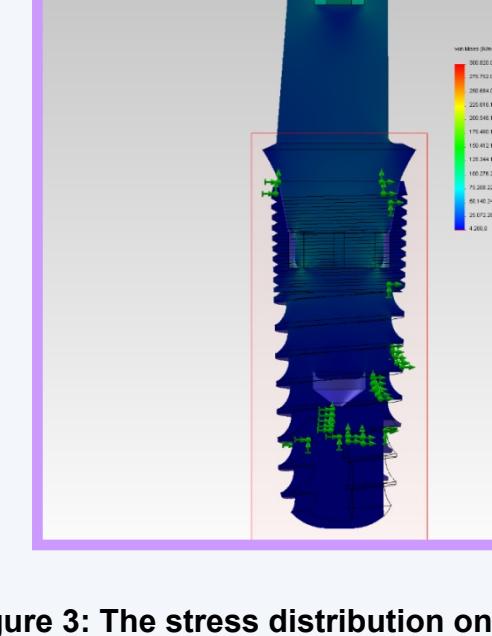


Figure 3: The stress distribution on Implant KA after applying angled 200 N force

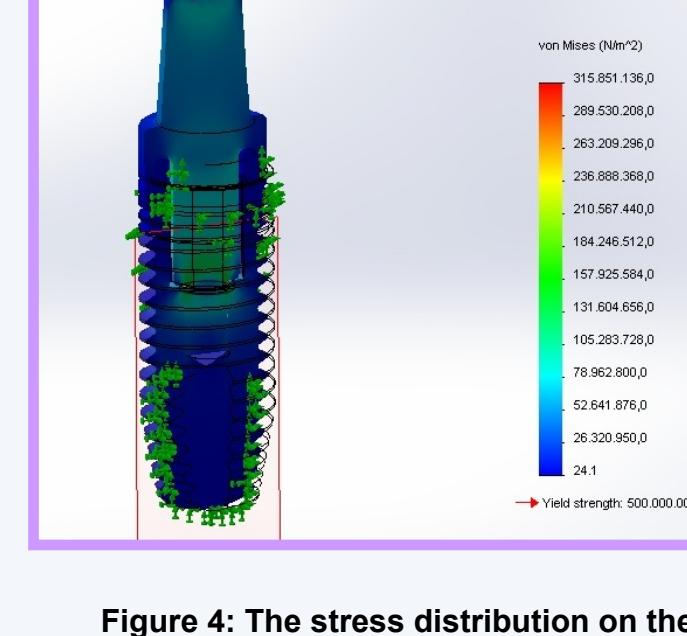


Figure 4: The stress distribution on the other implant after applying 300 N force

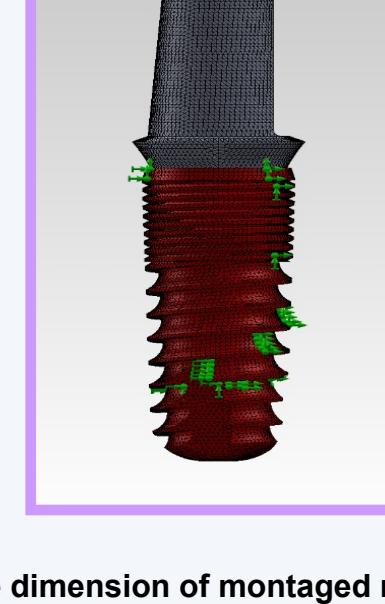


Figure 5: The dimension of mounted mesh on the other implant

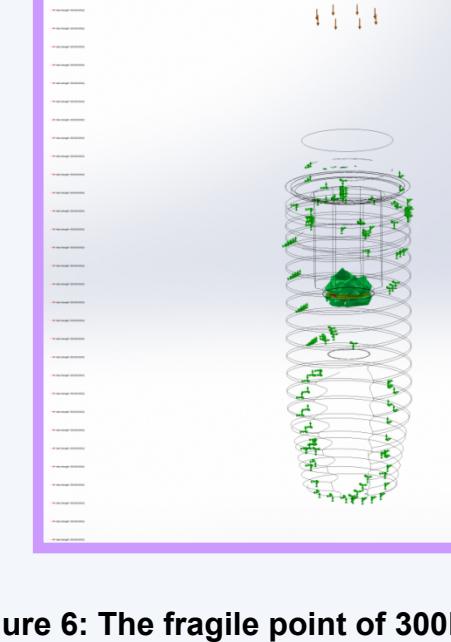


Figure 6: The fragile point of 300N mounted other implant

## Conclusions

Since the highest stress was observed at implant neck, the design of an internal connection, that distributes the forces homogeneously around the implant collar, is essential. Analysis by finite elements was shown to be a versatile and promising methodology for analyzing stress concentrations in implant dentistry, but it is worth emphasizing that the FEA (Finite Element Analysis) is an approximate virtual simulation of clinical situations presenting certain limitations.