10.5005/jp-journals-10024-1044
ORIGINAL RESEARCH


# Finite Element Analysis of Dental Implant as Orthodontic Anchorage

**Anirban Sarmah, Anirudh K Mathur, Vikas Gupta, Vinaya S Pai, S Nandini**

## ABSTRACT

**Aim:** The purpose of this three-dimensional (3D) finite element study was to investigate orthodontic loading simulation on a single endosseous implant and its surrounding osseous structure, to analyze the resultant stresses and to identify the changes in the bone adjacent to the implant following orthodontic loading.

**Materials and methods:** Two models were constructed using finite element method consisting of endosseous dental implant and the surrounding bone. In the first model, the contact between the implant and the bone was simulated showing no osseointegration, while the second model showed 100% osseointegration. Simulated horizontal loads of 20 N, at 90° from the long axis, were applied to the top of the implant. The study simulated loads in a horizontal direction, similar to a distal-mesial orthodontic movement.

**Results:** In the first model, the stress was mainly concentrated at the neck of the implant and at the closest surrounding bone. In the second model, the stress was chiefly concentrated at the neck of the implant at the level of the cortical superficial bone. The stresses decreased in the cancellous bone area. On the implant, the highest stress concentration was at the first cervical thread decreasing uniformly to the apex. The stress distribution on the mesial and distal sides showed that the maximum compressive stress was localized mesially and the maximum tensile stress distally. If both models are compared, it can be observed that the stresses were less and more evenly distributed in model 1 (initial stability) than in model 2 when osseointegration was assumed.

**Conclusion:** A lack of bony support for the implant represents an unfavorable situation from biomechanical point of view that should be considered and solved. As clinical problems mostly occur at the marginal bone region (bacterial plaque accumulation, overcontoured abutments, infections, osseous defects), attention should be focused on this region.

**Clinical significance:** When osseointegrated implants are primarily used as anchorage for orthodontic purposes and then as fixed prosthesis, the functional and structural union of titanium to bone should be preserved.

**Keywords:** Three-dimensional model, Implant, Mechanical forces, Stress, Finite element method.

**How to cite this article:** Sarmah A, Mathur AK, Gupta V, Pai VS, Nandini S. Finite Element Analysis of Dental Implant as Orthodontic Anchorage. J Contemp Dent Pract 2011;12(4):259-264.

**Source of support:** Nil

**Conflict of interest:** None declared

## INTRODUCTION

Orthodontics is gradually changing from an opinion-based practice to evidence-based practice. In contemporary period, it is necessary to have scientific rationale for any treatment modality and the evidence of tissue response to it. The greatest progress lies in perceiving some unifying concepts in the abundant evidence and ideas.

The number of adults seeking orthodontic treatment has increased significantly in recent decades.[1] However, not everyone has adequate dentition for orthodontic anchorage, e.g. partially edentulous patients and those with congenital dentofacial anomalies.[2] So, these patients often require multidisciplinary treatment approach. Anchorage control can be one of the nightmares for an orthodontist especially when they are restoratively or periodontally compromised or having multiple tooth agenesis.[1] Implants are an excellent alternative to traditional orthodontic anchorage methodologies, and they are a necessity when dental elements lack quantity or quality, when extraoral devices are impractical or when noncompliance during treatment is likely.

Stress analysis of dental implants is necessary for the investigation of bone turnover and maximum anchorage success. Incorrect loading or overloading may lead to disturbed bone turnover and consequent implant loss.[3] Since clinical determination of stress and strain distribution in the bone is not possible, an alternative technique should be used.

Detailed information to assess the stress and strain around a dental implant is difficult to obtain and analyze by other experiments. Hence, finite element analysis is used as this is the most reliable method.

The aims and objectives of the present study are:

1. To investigate orthodontic loading simulation on a single endosseous implant and its surrounding osseous structure.
2. To analyze the resultant stresses.
3. To identify the changes in the bone adjacent to the implant following orthodontic loading.

## MATERIALS AND METHODS

In this study, a three-dimensional finite element model of an endosseous implant and its surrounding structure was generated and used to analyze the resultant stresses generated and also to identify the changes in the bone adjacent to the implant following orthodontic loading.

### Steps Involved in the Generation of Finite Element Model

- Three-dimensional finite element model was constructed after examining the implant with and without osseointegration, consisting of the endosseous dental implant and the surrounding bone.
- The implant used in the study is a threaded endosseous implant made of commercially pure titanium of 4.2 mm diameter and 10 mm length manufactured by HITEC, Israel.

### Modeling the Implant and the Surrounding Bone

- The modeling was done using the software ANSYS 10
- Once the dimensions of the bone and the implant were obtained, these values were fed as input in both x and y dimensions into the modeling software
- These points were joined by lines to create the 2D cross-section of the implant and the bone
- Then, this cross-section was revolved 360° to get a 3D model
- The final model had 10213 tetrahedral elements and 2492 nodes.

### Assigning the Material Properties

In this study, the assumption was made that the materials were homogeneous and isotropic and that they had elastic material behavior characterized by two material constants *viz* Young's modulus and Poisson's ratio. Young's modulus and Poisson's ratio of implant (titanium), compact and cancellous bone are used as per the literature.

| *Material* | *Young's modulus (1 Mpa = $10^6$ Pa)* | *Poisson's ratio* |
|---|---|---|
| Compact bone | 13760 MPa | 0.30 |
| Cancellous bone | 7930 Mpa | 0.30 |
| Implant (titanium) | 110000 Mpa | 0.35 |

### The Finite Element Model

Two types of finite element models of the implant-bone complex were generated. These are:

- In the first model, the contact between the implant and the bone was simulated. The materials were elastic and it was assumed that there was no osseointegration. Thus, a contact condition needed to be imposed with no penetration of one material into other and with friction ignored. This model configuration represented the situation immediately after implantation when the implant was totally surrounded by cancellous bone.
- In the second model, it was assumed that the material was elastic and that 100% osseointegration was complete. The classic finite element model in elasticity could be considered. After osseointegration, there was no difference at the contact boundary between the surrounding bone and the implant. Boundary nodes of both parts were designed to be common and so it could be assumed that the complex bone implant was a unique domain composed of two mechanical parts (each of them with different elastic coefficients); the surrounding bone and implant.

### Implant Loading

- Simulated horizontal loads of 20 N, at 90° from the long axis, were applied to the top of the implant. The study simulated loads in a horizontal direction, similar to a distal-mesial orthodontic movement.
- Stresses (in MPa) were calculated and presented as colored contour bands; different colors representing different stress levels in the deformed state. Positive or negative values of the stress spectrum indicate tension and compression respectively.

## RESULTS

### Interpretation of Results

The results obtained are discussed based on the resultant stresses on the implant and the bone adjacent to the implant in both the models. The distribution of maximum and minimum principle stresses and strain throughout the various components of the model *viz* implant and bone adjacent to the implant are presented in the form of color bands. The legends below the screen indicate the magnitude of stress depicted by each color. The colors on the right side indicate tensile stress and colors on the left side indicate compressive stress. Colors on the extreme right and on the left side, that is, the red and the blue color indicate the highest magnitude of tensile and compressive stress respectively with gradual decrease of the magnitude of stress depicted by other colors. Positive values indicate tensile stress and negative values indicate compressive stress (Fig. 1).

**Fig. 1:** Element of implant-bone complex

For the sake of better understanding, the following terms will be used: The surface of the implant on which the load is applied is taken as distal, the opposite surface is mesial. The other terms are head and tip of the implant.

Two types of stresses were considered for this study. These were the maximum principle and minimum principle stresses. In engineering theory, stresses act on a point in more than one plane. For appropriate study of the stresses in a solid, two mutually perpendicular directions of stress must be studied. These two directions are represented by maximum and minimum principle stresses and may not correspond to the direction of any of the axes. It is possible for a solid to experience tensile stresses in one plane and compressive stresses in the mutually perpendicular plane. Hence, in a single area, maximum principle stresses are tensile and minimum principle stresses are compressive (Fig. 2).

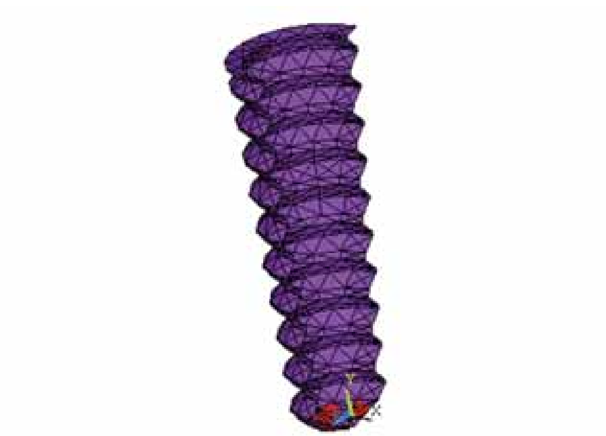


**Fig. 2:** Elements of the implant model

## Model 1 (Without Osseointegration) (Figs 3A to D)

### *Distribution of Stresses on the Implant (Figs 3A and B)*

On the distal surface, highest magnitude of tensile stress was seen at the upper half of the implant and highest magnitude of compressive stress was seen at the apex.

On the mesial surface, highest magnitude of compressive stress was seen at the neck of the implant which gradually decreases at the center of the implant and highest magnitude of tensile stress was seen at the apex.

### *Distribution of Stresses on the Distal Surface of the Cancellous Bone (Fig. 3C)*

On the distal surface, there is a uniform distribution of stresses in the cancellous bone around the implant with high concentration of tensile stress near the neck of the implant and compressive stresses near the tip of the implant.

### *Distribution of Stresses on the Mesial Surface of the Cancellous Bone (Fig. 3D)*

On the mesial surface also, there is a uniform distribution of stresses in the cancellous bone around the implant with high concentration of compressive stress near the neck and tip of the implant.

## Model 2 (With 100% Osseointegration) (Figs 4A to C)

### *Distribution of Stress in the Implant (Fig. 4A)*

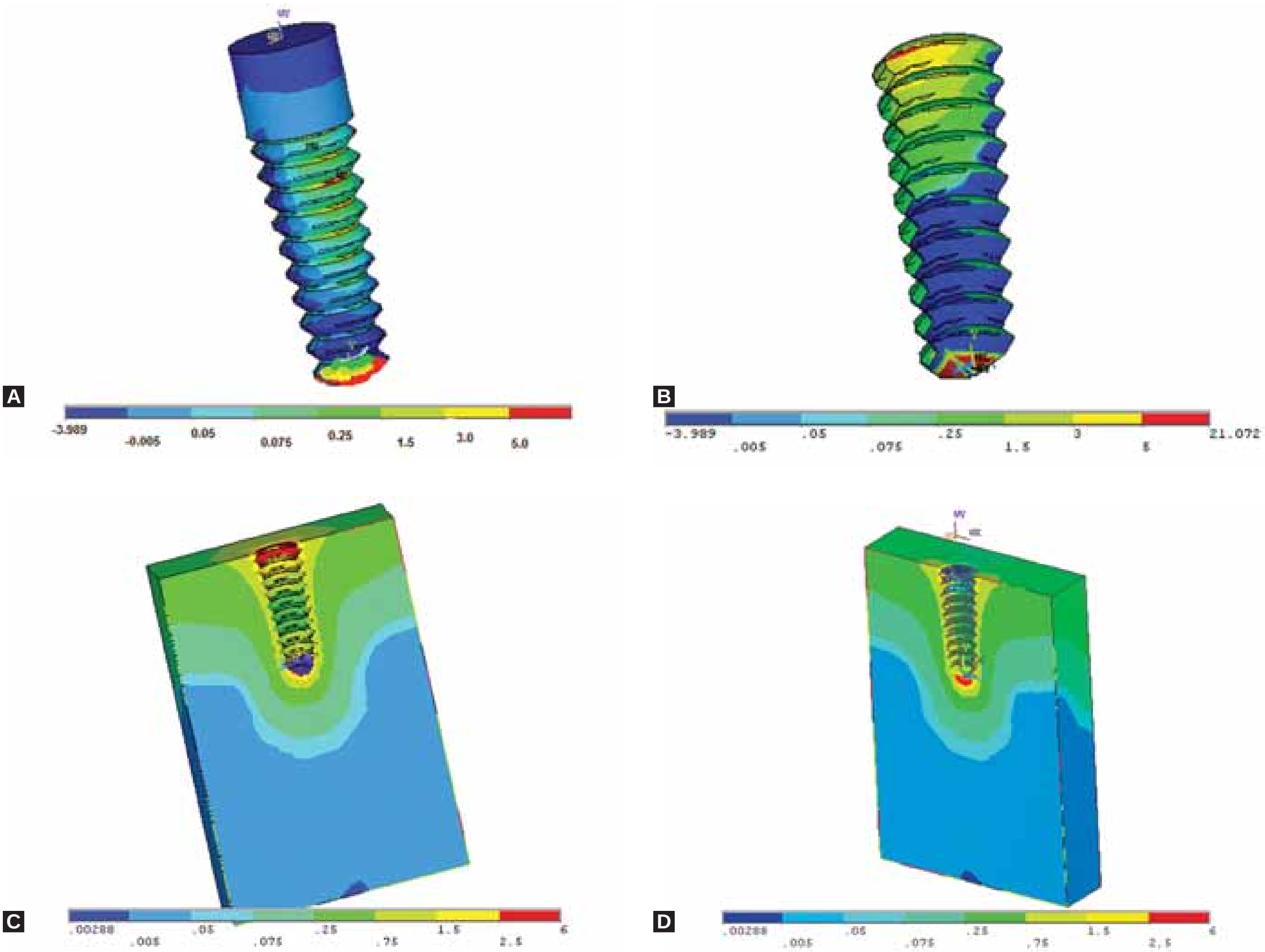
In the implant, compressive stresses were seen on the mesial surface with the maximum compressive stress being concentrated at the neck of the implant. The maximum tensile stresses were concentrated at the point of force application. The compressive stresses were greater than the tensile stresses. These compressive and tensile stresses gradually reduced toward the tip of the implant. But, the tip of the implant experienced a tensile stress when compared to the bulk of the implant, which experienced more of compressive stress.

### *Distribution of Stresses on the Cortical Bone (Fig. 4B)*

The stresses were concentrated chiefly at the neck of the implant and at the level of the cortical superficial bone. The cortical bone experienced lower compressive and tensile stresses, but there were no significant differences between the tensile and the compressive stresses in the bone.

### *Distribution of Stresses in the Cancellous Bone (Fig. 4C)*

The cancellous bone experienced lower stresses when compared to the cortical bone. But, in contrast to the cortical bone, compressive stresses were slightly greater than the tensile stresses. Within the cancellous bone, maximum stress concentration of both tensile and compressive stresses was seen at the corticocancellous junction.




**Figs 3A to D:** Model 1 (without osseointegration): (A) Distal aspect of implant; (B) Mesial aspect of implant; (C) Distribution of stresses on the distal surface of the cancellous bone; and (D) Distribution of stresses on the mesial surface of the cancellous bone

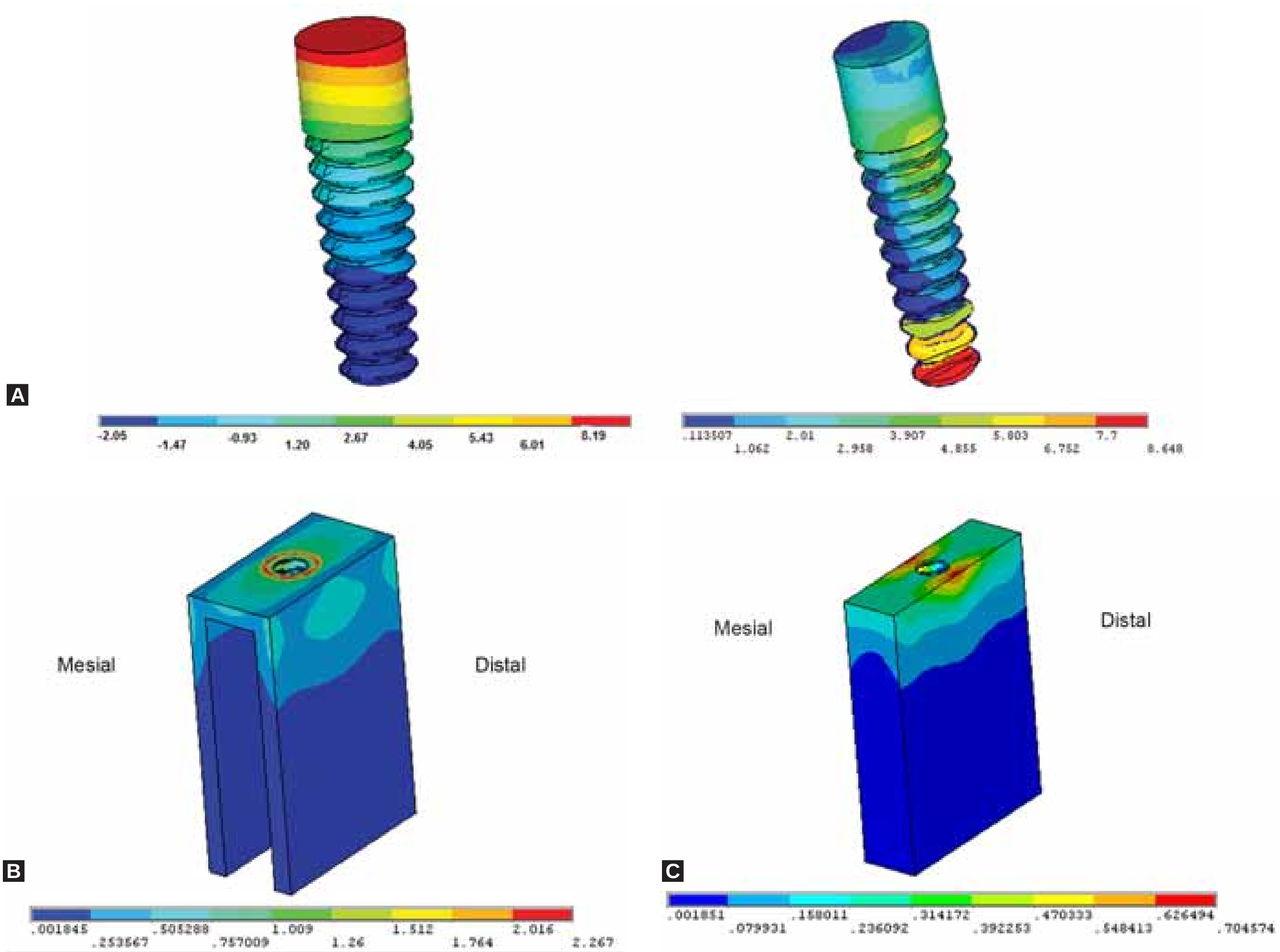
If both models are compared, it can be observed that the stresses were less and more evenly distributed in model 1 (initial stability) than in model 2 when osseointegration was assumed.

## DISCUSSION

Gallas et al[1] constructed two three-dimensional models of bone and implant complex and analyzed the stress distribution. Implant used was 4.1 mm in diameter and 10 mm in length. The first model assumed that there was no osseointegration whereas the second model had 100% osseointegration. A 20 N of horizontal load was applied at 90° to the long axis of the implant. The results indicated that the maximum stresses were located around the neck of the implant, in the marginal bone.

Vasquez et al[4] constructed an endosseous implant and an upper canine with its periodontal ligament and cortical and cancellous bone. Levels of initial stress were measured during two types of canine retraction mechanics (friction and frictionless). Von Mises stresses in the evaluated loads showed the highest stresses on the implant and the cortical bone at the cervical third. The lowest stresses appeared at the apical third of the implant. The highest stress was observed in the implant, followed by the cuspid, the cortical bone and finally the periodontal ligament.

Chen et al[5] compared the anchorage effects of different palatal osseointegrated implants using finite element analysis. Three types of cylinder implants were investigated. Each consisted of two maxillary second premolars, their associated periodontal ligament and alveolar bones, palatal bone, palatal implant and a transpalatal arch. Another model without an implant was used for comparison. Horizontal force (mesial 5 N, palatal 1 N) was loaded at the buccal bracket of each second premolar, and stress in the periodontal ligament, implant and surrounding bone was calculated. The stress distribution on the implant and bone complex showed that stress declined steadily from the cervical part to the apex.




**Figs 4A to C:** Model 2 (with 100% osseointegration): (A) Distribution of stress in the implant; (B) Distribution of stresses on the cortical bone; (C) Distribution of stresses in the cancellous bone

However, using the finite element method it was found that the highest risk of bone resorption occurs at the neck region of an implant. The stress distribution was less concentrated and more uniformly distributed at the neck region of the first (initial stability) than of the second (osseointegration) model because of a different biological adaptation to loads (bone elasticity *vs* formation of osseous union).

It is important to note that osseointegrated implants are able to support orthodontic loading and may function as adequate anchorage units. It is therefore very important not to jeopardize the bone-implant interface with traumatic loading situations.[6] The results of the present study illustrate that there is a greater risk of overload at the mesial and distal bone. This should be taken into account in patients where a narrow alveolar bone ridge exists, as in some adult patients with several missing posterior teeth where an endosseous implant is being used for orthodontic anchorage.

Because orthodontic loading does not necessarily mean that the ultimate strength of bone tissue will be exceeded, continuous loading is more likely to cause fatigue damage (bone microcracks, marginal bone resorption) that could jeopardize the anchorage unit. From a mechanical point of view, the presence of bone defects seems unfavorable due to the lack of bone support. Conversely, periimplant bone stresses and strains are not only a function of the *in vivo* loading conditions, but are also determined by the bone quality (bone mechanical properties) and quantity (cortical bone thickness, cancellous bone density), periodontal status, oral hygiene and numerous other factors that may play a role in marginal bone remodeling.[7]

## CONCLUSION

When osseointegrated implants are primarily used as anchorage for orthodontic purposes and then as fixed prosthesis, the functional and structural union of titanium to bone should be preserved. A lack of bony support for the implant represents an unfavorable situation from biomechanical point of view that should be considered and solved. As clinical problems mostly occur at the marginal bone region (bacterial plaque accumulation, overcontoured abutments, infections, osseous defects), attention should be focused on this region.

## CLINICAL SIGNIFICANCE

In the implant, the most critical area is its neck, where there is maximum stress concentration, and the marginal bone (cervical margin) which surrounds it. Thus, this area should be preserved clinically in order to maintain the bone-implant interface structurally and functionally.

It was seen that the implant tipped to a very negligible amount in the direction of the load applied, like a tooth tipping on application of load. But the displacement seen was very negligible and clinically insignificant.

Based on the experience from our study, the following suggestions can be made for optimization of the implant design:

- The neck of the implant must be long enough to project away from the soft tissues, so that any attachments placed on the implants do not impinge on the mucosa. The inflammation of the overlying soft tissue and/or the marginal bone resorption can jeopardize the stability of the implant.
- The neck of the implant must be sturdy enough, since the maximum stress concentration occurs at the neck of the implant. If the implant is not strong in this region, it may affect the integrity of the implant.
- When using the implant for orthodontic loading, it is advisable to take all the necessary precautions to place the implant as much in the cortical bone as possible. The reason is that the stress and strain values in the trabecular bone were very low, which would result in atrophy of the surrounding bone.

## REFERENCES


1. Gallas MM, Abeleira MT. Three-dimensional numerical simulation of dental implants as orthodontic anchorage. Eur J Orthod 2005;27:12-16.
2. Huang KH, Shotwell JL. Dental implants for orthodontic anchorage. Am J Orthod Dentofacial Orthop 2005;127:713-22.
3. Gedrange T, Bourauel C. Three-dimensional analysis of endosseous palatal implants and bones after vertical, horizontal, and diagonal force application. Eur J Orthod 2003;25:109-15.
4. Vasquez M, Calao E. Initial stress differences between sliding and sectional mechanics with an endosseous implant as anchorage: A three-dimensional finite element analysis. Angle Orthod 2001;71:247-56.
5. Chen F, Terada K. Anchorage effects of palatal osseointegration implant with different fixation: A finite element study. Angle Orthod 2005;75:593-601.
6. Rubo JH, Souza EAC. Finite element analysis of stress in bone adjacent to dental implants. J Oral Impl 2008;34:248-55.
7. Zhang Y, Zhang D, Feng C, Peng P. A three-dimensional finite element analysis for the biomechanical characteristics of orthodontic anchorage microimplant. J Hard Tissue Biology 2006;15:69-72.


## ABOUT THE AUTHORS


### Anirban Sarmah (Corresponding Author)

Chief Orthodontist, Department of Orthodontics, New Dental Clinic, Jorhat, Assam, India, Phone: +91-9678010720, e-mail: anirbansarmah@yahoo.com

### Anirudh K Mathur

Assistant Professor, Department of Orthodontics, HKDET Dental College, Humnabad, Karnataka, India

### Vikas Gupta

Assistant Professor, Department of Orthodontics, JCDV Dental College, Sirsa, Haryana, India

### Vinaya S Pai

Professor and Head, Department of Orthodontics, Bangalore Institute of Dental Sciences, Bengaluru, Karnataka, India

### S Nandini

Professor, Department of Orthodontics, Bangalore Institute of Dental Sciences, Bengaluru, Karnataka, India