Stress Distribution of a New Type of Perforated Implants in Femur Neck Fracture: Finite Element Model

Afgan Jafarov¹, Chingiz Ali-Zadeh², Zafer Özer³, Amirullah M. Mamedov⁴, Ekmel Özbay⁴

¹Modern Hospital, Baku, Azerbaijan
²Azerbaijan Scientific-Research Institute of Traumatology and Orthopedics, Baku, Azerbaijan
³Mersin University, Mersin Vocational High School Electronic and Automation Department, Mersin, 33335, Turkey
⁴Bilkent University, Nanotechnology Research Center (NANOTAM), Ankara, 06800, Turkey
⁵International Scientific Center, Baku State University

E-mail: mamedov46@gmail.com

Abstract. Osteosynthesis of femoral neck fractures (FNF) is accompanied by a large percentage of various complications. The aim of this research was to study biomechanical implications arising from the stress and strain distributions by use of new device – perforated H-beam implant. A reference 3D femur model has been developed using digital femur geometry. The stress and deformity characteristics after the application of force were determined by ANSYS, which entails the finite element analysis (FEA) method. This method was used for both the perforated and non-perforated implant models and compared the application of constant strength. Three types of FNF were realized according to the Pauwell classification system.

We concluded that, the perforations in the implants make the distribution of strength and pressure homogenous along the whole implant functioning as wave breaker to reduce the pressure on the bone. This contributes to implant stability and the minimization of bone destruction.

1. Introduction

Scientists and clinicians have long been interested in the assessment of the mechanical behavior of human whole bones. The general impetus has been twofold, namely to understand the various geometric and material factors that affect gross mechanical behavior under various physiological loading conditions and to determine which orthopedic repair techniques and implants can best preserve or improve intact whole bone biomechanical strength and stability. To these ends, both experimental methodological and theoretical models have been reported in the literature [1-3]. For example, in the assessment of the performance of orthopedic implants to identify areas of high stress, and to facilitate implant design, biomechanical testing has often been augmented by the use of finite element models (FEM) of bone-implant systems.

Finite element analysis of a whole bone was first performed in [1-2] to gain a perspective on the stresses encountered by the human femur (2D geometry), although later models were 3D in nature [3].
Whole bone FEM studies of these types ignore the structure of bone, instead treating it as continuum. Nevertheless, this approach is useful for implant design purposes, where comparative, rather than absolute, values of stress are required. More fundamental properties, such as the microstructure, loading mode, and elastic properties of tissue have been obtained using FEM. On the other hand, the most important problem for elderly patients is the problem of the implant’s internal fixation method and its geometry.

The aim of the present investigation was to gain insight into the biomechanical implication arising from the stress and strain distributions induced by the implantation of specific fixation devices: perforated and non-perforated H-beam implants. The FEM that has been accepted as a standard tool for the analysis of a wide range of biomechanical problems since its introduction into this research field during the early 1990s was used to investigate the stress in the implants and the surrounding bone material under typical loading conditions. These stress patterns were compared with those found in a physiological uninjured femur, exposed to the same loading environment (Fig. 1)

1.1 Model Geometry, Implants, and Fracture zone

A major simplification of the essentially 3D nature of the problem nature of the problem to be studied can be based on the observation that the external forces corresponding to appropriate representative load systems (such as the single leg stance phase of normal gait) are acting mainly in the mid frontal plane. Accordingly, the relevant mechanical responses of the bone-implant configurations are dominated by in plane deformations and stresses. Thus, provided that the tensile, compressive, shear and bending stiffnesses of the bone-implant system (Fig. 3) are adequately represented, two-dimensional plane stress FE-models are sufficient for investigating this problem. Specific layers of elements for each constituent of the femur-implant system (i.e. steel implant, cortical bone and trabecular bone) were introduced. The thickness of each individual finite element being selected according to the real geometry of the implants and a representative femur, taking into account the medullary cavity of the femoral shaft, and the reduction of bone material thickness caused by implantation as well as the percentage of cancellous and cortical bone in the femoral head. By using these overlay techniques, the numerical requirements were significantly reduced without seriously compromising the realism of the models.

Therefore, the objectives of this study are as follows:

1. To examine the perforated and non-perforated H-beam implants (see, Fig. 3)
2. Investigate the physico-mechanical behavior of the H-beam implants using the finite element method.
3. To examine various stress, strain, and changes in the geometry of the different implant groups.

Three osteotomies through the trochanteric region, inclined at an angle of 30°, 50°, and 70° respectively, formed the quasi-real model of an unstable trochanteric fracture. Three variations of this idealized wedge shape fracture were used to model clinically relevant configurations. This configuration was approximated by reducing the fracture area to a zigzag line, along which frictionless contact between the surfaces was assumed, resulting in a non-linear contact problem. In a second scenario, a callous bridging was assumed, filling the wedge-shaped fracture zone in the cancellous as well as in the cortical layer with callus of relatively low stiffness. The third configuration addressed the fully healed state of the femur with completed bony bridging.

2. The model and calculation method

The calculations of the stress and shape change condition in the implant were carried out by the finite element method. The newly developed hip prosthesis, shown in Figure 3, was investigated for mechanical behavior in the modeled femur bone, the Pauwel's classification system (PCS) type1 in Figure 1, type 2 and type 3 fractures.

The key points used in the reference model of the femur bone were obtained from the two-dimensional digital picture in the CAD environment. When creating a 3D model, the bottom-up
modeling approach was combined with key points splines, fields using splines, and volumes were created using fields.

![Figure 1. Pauwel's classification system (PCS-femur bone)](image)
a) Type 1, b) Type 2 c) Type 3

The results of the surgical titanium prosthesis were applied to the femur and the result was evaluated by the finite element method, with a vertical force of up to 5 times the weight of a normal weight person during walking. The anatomical femur was investigated using a three-dimensional finite element model (3D-FEA). The measurements of the 2-dimensional femur bone for modeling were transferred to CAD software.

The 4000 N force, which occurred during the walking movement of a person of normal weight and up to 5 times its weight, was applied to 4 different points in 1000N. This is based on one leg of the press during the walk, and the load transferred from the hip joint to the femur head sphere was applied in accordance with the mechanical axis. The total number of elements and nodes in the implanted models are as shown in Table 1. The material properties used in the analysis were taken from the literature [17]. Figure 2 shows the applied force and boundary conditions on the femur. For FEA, the Solid 92 element in a 3-D 10-node tetrahedral structure (triangular finite elements with six nodes and semi-circular side were used) suitable for dividing irregular structures such as the hipbone into meshes was used.

![Figure 2. Directions of applied forces and boundary conditions](image)
3. Results and Discussions

According to the results obtained, the amount of Von Mises stress consisting of both models (hole-free and perforated) in parallel to the increase of the fracture type from 1 to 3 as increased. Maximal stress is increasing in the case of cortical and trabecular bone models with the increase of the non-hanging angle of the model. Only in Model 2/Type 2 class Q types, changing is calls in the maximal stress of cortical bone (Tables 2-3).

In all fracture types of Model 1, the amount of stress falling on the cortical bone of the maximal Von Mises that fall into the trabecular bone region was observed at an average of 1/3 percent. Model 2/Type 1 fracture type also continues to 1/3 ratio. However, in the Model 2/Type 2 and type 3 fracture types, the maximal ratio of trabecular and cortical bone weaving was observed in 1/2. These results indicate that there are holes in the implants and the bathtubs that fall into the trabecular bone models even increase. In Model 1 and 2 samples, in comparison with type 3 fractures, the most significant increase in stress figures on the implant and trabecular bone tissue was calculated (521.324/285.339 on the implant; 58,111/32,056) on the trabecular tissue. In both cases, there was a 1.8 fold increase. In the fracture area of Pauwel’s Type 3 fractures, the implant and trabecular bone area were observed to decrease the forces. Tables 2-3 summarize the numerical results for two types of implants. A comparative analysis of the results obtained by using two types of implants showed a good convergence deviations ranging in reasonable limits. The plane model is suitable for carrying out a more detailed study of the state of stresses, using finite element analysis, behavior under mechanical load for proximal area. As a result of the action of mechanical strain forces in the femoral head and the greater trochanter in the elements that compose the structure is generated axial efforts (tensile and compression). Therefore, by way of finite element analysis, the state of stresses identifying the
maximum loaded area can be shown. To characterize this state of stresses, Von-Mises equivalent stress distributions are summarized in table 2. Von-Mises equivalent stress values and the maximum and minimum principal stresses depending on application force F.

Trabecular structure is shown taking over the mechanical load and is distributed to the cortical structure and especially toward diaphysis where the maximum stress is registered, the critical area – femoral neck – thereby subjected to a lower load level. It is noted as an important role that it plays in the trabecular structure and integrity of this structure depending on the integrity of the structure of the human femoral bone (proximal area).

| Table 2. Von Mises stress distribution on Model 1 implant. |
|----------------------------------------------------------|
| Maximum Von Mises Stress [MPa]                            |
| Type1 | Type2 | Type3 |
| Prosthesis | 210.673 | 278.530 | 285.339 |
| Trabecular Bone | 23.949 | 31.281 | 32.056 |
| Cortical Bone | 70.630 | 90.094 | 95.377 |

| Table 3. Von Mises stress distribution on Model 2 implant. |
|----------------------------------------------------------|
| Maximum Von Mises Stress [MPa]                            |
| Type1 | Type2 | Type3 |
| Prosthesis | 209.821 | 300.996 | 521.324 |
| Trabecular Bone | 23.915 | 33.705 | 58.111 |
| Cortical Bone | 70.392 | 67.117 | 116.012 |

In the Model 1 and 2 samples, in comparison with type 3 fractures, the most significant increase in stress figures on the implant and trabecular bone tissue was calculated (521.324/285.339 on the implant); 58.111/32.056) on the trabecular tissue. In both cases there is a relatively 1.8 fold increase. After all, in the fracture area of Pauwel’s Type 3 fractures, the implant and trabecular bone area was observed to decrease the forces.

4. Conclusion

The finite element method is an easy and safe method to describe the weight of the hips of the hip play, the geometry of the ink in the different phases of the surface, and the load distribution on the implants used in different fracture types [18-19].

In the computer environment, it is easier to perform stress analysis with FEA and predict errors for the prostheses and implants designed. FEA was initially developed for the aircraft industry, and it was also used in the analysis of implants developed in medicine over time. 3D-FEA allows us to understand the biometric properties of implants better, thereby helping the designer in the design and improvement of the material used in the implant biomechanical functions.

In our study, a finite element method was used to investigate the load distribution around bone tissue and implant with holes on the new implant that were planned for osteosynthesis of femur neck fractures. The findings of the perforated and hollow implant models were compared and different and interesting results were obtained. After loading in perforated implant models, more strain was observed on the implant. This strain also affects the trabecular tissue regions.

References

[1] Ström O, Borgstrom F, Kanis J A, Compston J, Cooper C, McCloskey E V, and Jonsson B 2011. Arch. Osteoporos 6 59
[2] Raaymakers E L 2006 Acta Chir. Orthop. Traumatol. Cech. 73 45
[3] Borqvist L, Lindelöw G, and Thorngren K G 1991 Acta orthop. Scand 62 39
[4] Lagerby M, Asplund S, and Ringqvist I 1998 Acta Orthop. Scand. 69 (4) 387
[5] Khoo CCH 2014 Malays Orthop. J. 8 (2) 14 PMCID: PMC4181088
[6] Osarumwense D, Tissuing E, Wartenberg K, Aggarwal S, Ismail F, Orakwe S, and Khan F 2015 Clin. Orthop. Surg. 7 (1) 22
[7] Boraiah S, Paul O, Gardner MJ, Parker RJ, Barker JU, Helfet D, and Lorich D 2010 Arch. Orthop. Trauma Surg. 130 (12) 1523
[8] Shields E and Kates S L 2014 Arch. Orthop. Trauma Surg. 134 1667
[9] Gjertsen J E, Vinge T, Engesaeter L B, Lie S A, Havelin L I, Furnes O, and Fevang J M 2010 J. Bone Joint Surg. Am. 92 (3) 619
[10] Wild M, Jungbluth P, Thelen S, Laffrée Q, Gehrmann S, Betsch M, Windolf J, Hakimi M 2010 Orthopedics. 11 33
[11] Xu M, Zhang L, Mao Z, Wang H, Chen H, Guo Y, Tao S, Zhang Q, Liang X, Tang P 2010 Zhongguo Xiu Fu Chong Jian Wai Ke Zhi. 24 (12) 1419
[12] Bhandari M, Devereaux P J, Tornetta P, Swiontkowski M F, Berry D J, Haidukewych G, Schemitsch E H, Hanson B P, Koval K, Dirschl D, Leece P, Keel M, Petrisor B, Heetveld M, and Guyatt G H 2005 J. Bone Joint Surg. Am. 87 (9) 2122
[13] Strömqvist B, Hansson LI, Nilsson L T, and Thorngren K G 1987. Clin. Orthop. Relat. Res. 218 58
[14] Rehnberg L and Olerud C 1989 Acta Orthop Scand 60 (5) 579
[15] Lu-Yao G L, Keller R B, Littenberg B and Wennberg W E 1994 J Bone Joint Surg 76-A15–25
[16] Mattsson P Larsson S 2003 Scandinavian Journal of Surgery 92 215
[17] Atik F, Özkan A and Uygur I 2012 Sakarya Üniversitesi Fen Bilimleri Enstitüsü Dergisi 16 (3) 249
[18] Voo L, Armand M, and Kleinberger M. 2004 Johns Hopkins APL Technical Digest 25 223
[19] Bouguerara H, Bureau M N, and Yahia L 2010 J. Biomed. Mat. Res. Part A 92 164