



Mechanical Analysis of Bone-Plate Construct Regarding Strength and Stiffness

Rana Idan Abed^{1*}, Sadiq Jaafer², Waleed A Alsaadan³

Authors affiliations:

1*) Biomedical Engineering,
Al-Nahrain University,
Baghdad-Iraq.
ranaidan2@gmail.com

2) Biomedical Engineering,
Al-Nahrain University,
Baghdad-Iraq
sadiq_hamandi@yahoo.com

3) Consultant Orthopedic
Surgeon, Medical City,
Baghdad-Iraq.
waleedbgh@yahoo.com

Paper History:

Received: 10th Dec. 2019

Revised: 31th Dec. 2019

Accepted: 29th Jan. 2020

Abstract

The aim of this study was to support surgeons to decide where to place the screws in order to achieve an optimal fracture healing and to prevent implant failure after a femoral shaft fracture. So this paper focus on the analysis of bone-plate construct by using Finite element Analysis (FEA), comminuted femur fractured bone fixed with Dynamic Compression Plate (DCP) 16 holes by 4.5 Cortex screws, to investigate the effects of screws configuration on the mechanical behavior of different seven model as Interfragmentary strain which is the most important factor for femur fracture healing. The results state the relationships between the Von-Mises stress, Total deformation and Interfragmentary strain with respect to the screws configuration. The study shows the regions of maximum stress from stress distribution and also founded that we can decrease the Interfragmentary strain by increasing the number of screws.

Keywords: Bone Fixation, Dynamic Compression Plate, Interfragmentary Strain, Finite Element Analysis.

التحليل الميكانيكي لتثبيت عظم بصفيحة فيما يتعلق بالقوة والصلابة

رنا عيدان عبد، د. صادق جعفر عباس، د. وليد عبد الحسن السعدان

الخلاصة

الهدف من هذه الدراسة هو مساعدة الجراحين لتحديد مكان وضع البراغي من أجل تحقيق الشفاء الأمثل للكسر ومنع فشل الزرع بعد كسر عمود الفخذ لذا فإن هذا البحث يركز على التحليل الميكانيكي لنموذج العظم والصفيحة باستخدام تحليل العناصر المحددة. يثبت العظم المكسور بصفيحة ضغط ديناميكية بواسطة براغي لدراسة تأثير مواقع البراغي على التصرف الميكانيكي لسبعة نماذج مختلفة من ترتيب البراغي، كتوتر القطع الداخلية للكسر الذي هو عامل مهم في تحديد التنام الكسر. وضحت النتائج العلاقات بين كل من الإجهاد والتشوه وتوتر قطع الكسر المفتت مع ترتيب مواقع البراغي، كذلك بينت الدراسة مناطق الإجهاد العالي في توزيع الإجهادات ووجدت إمكانية تقليل توتر قطع الكسر المفتت عن طريق زيادة عدد البراغي المستخدمة في التثبيت

1. Introduction

Femur fractures are considered to be the most common fractures due to the maximum moment in the bone shaft. The most popular technique for fixing femoral fracture is the internal fixation [1].

The main concept of this study was treating the shaft of femur fractures by plates and screws. The task of the plates are getting the ends of the fractured bones with each other and fastening them with a metal plate by screws. The biggest defiance for the surgeons is the stable fixation because of the instability of the bones, as a result of that a high complexity rate because the screws became loose or the plate get broken [2].

ANSYS finite element analysis software involved in this study to test the standards of stress distribution and total deformation of bone-plate construct, by using a compression force as a static loading

conditions on a three dimensional model of bone-plate[3].

If the fracture takes place at the middle of the femur, the fracture would be cut and form a gap. The definition of interfragmentary strain (ϵ_{IF}) is the rate of deflection of the gap after applying force on the model and the original length of the gap as shown in Fig.1.

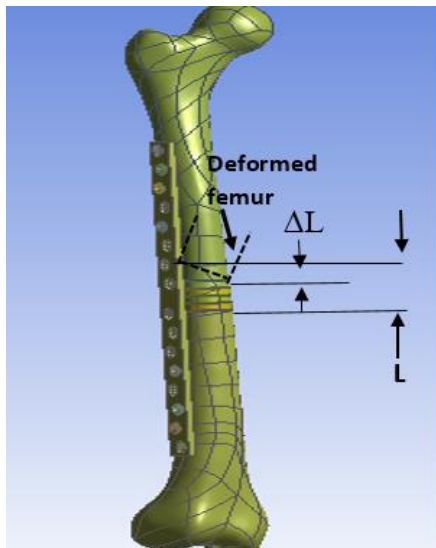


Figure (1): Fractured femur deformation

The equation of interfragmentary strain is:

$$\epsilon_{IF} = \frac{\Delta L}{L}$$

Where:

ΔL = the deflection of the gap after the applying force.

L = the original length of the gap.

The best Interfragmentary strain ranges from 2-10% (Perren, 1979; Kim et al., 2010) [4].

2. Method

Finite Element method

FEA is used to evaluate the mechanical stiffness and stability of the DCP–bone system with comminuted fracture as affected by differences of working length by varying screws positions under compression load [5].

2.1 Model Design

The geometry of the required 3D model is the first stage. The DCP of 16 holes is placed in touch to femur bone by 4.5 cortex as shown in Figures 1&2 [4]. The anatomical geometry of the long bone is assigned with length of 460 mm, an outer diameter of 25 mm, diameter of femoral head of 48 mm with intramedullary canal diameter of 12 mm and length of 366 mm, and the cortex screw with length of 28 mm and thread diameter of 4.5 mm, core diameter 3 mm [1].

The 3D solid models of these implants are created by CAD software (SOLIDWORK 2018). After that the bone-plate construct was gated by making assembly from gathering the particular components of the bone, plate and screws.

2.2 The Screw Configurations

The screws were placed according to different seven orientations stated in table 1 and the holes are assigned as numbers to state the screw place in models as in Fig.2 for example in model1 two screws used at 1st and 7th holes and so on the other models fixed. The holes which carry the number 8&9 not used. The number of screws used varies from 2-14 [1].

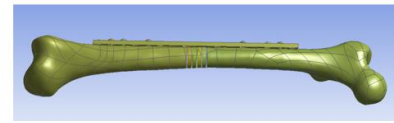
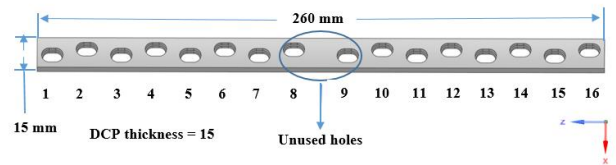


Figure (2): Screws holes number

Table (1): The screw configurations on each side

Model	Fixed at hole-number	Number of screws
1	1 16	2
2	1 3 14 16	4
3	1 2 3 14 15 16	6
4	1 2 3 5 12 14 15 16	8
5	1 2 3 4 5 12 13 14 15 16	10
6	1 2 3 4 5 7 8 10 11 13 14 15	12
7	1 2 3 4 5 6 7 10 11 12 13 14 15 16	14

2.3 Material Assignments

It is very complicate to allocate bone properties along each direction of the model because of the human bone highly heterogeneity and nonlinearity, so the femur bone considered as isotropic because the complete anatomical femur bone concerned in the analysis. Material properties are assigned in ANSYS engineering data [6].

The Young's modulus of the femur range from 10 to 20 GPa, so is taken to be 15 GPa. Density and Poisson ratio used are 2000 kg/m³, 0.3 respectively [7].

And the plate and screw materials used are stainless steel 316L and its properties as in Table 2 [8].

Table (2): Stainless steel 316L mechanical properties

Material	Density (kg/m ³)	Poisson ratio
S.S316L	8000	0.3

2.4 Importing the 3D models

The design was imported to ANSYS 2019.R1 Workbench geometry. Bonded contacts are formed directly by the workbench module between the adjacent surfaces of the assembly. With frictional contacts are inserted for plate to bone surfaces with coefficient of friction μ 0.2 [9].

2.5 Meshing

Mesh is generated by ANSYS Workbench 19.0 software for the femur, plate and screws as shown in Fig.3.

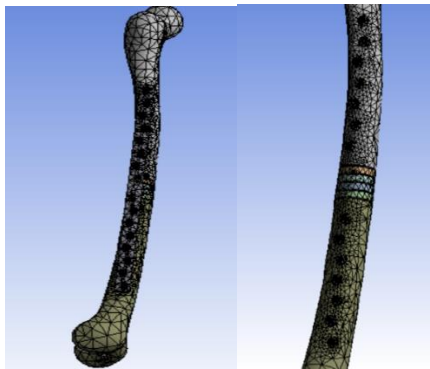


Figure (3): Model 7 meshing with ANSYS Workbench

Identical size and shape of elements are involved in meshing with frictional contact sizing at 3 mm element size. For example model 7 the number of nodes and elements were 321332 and 188751 for comminuted fracture model [9].

2.6 Boundary Condition

In static structural part, a fixed support is defined at the lower surface of the femur bone as shown in Fig.4 [10]. Compression force of 750N is marked at the femoral head surfaces based on the presumption that the person weight of 75 kg as shown in Fig.5 [6].

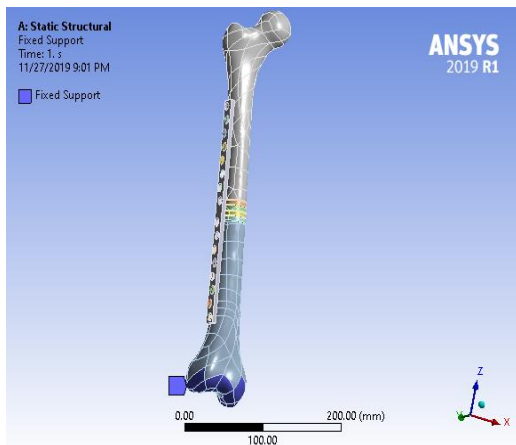


Figure (4): fixed support at femur lower end

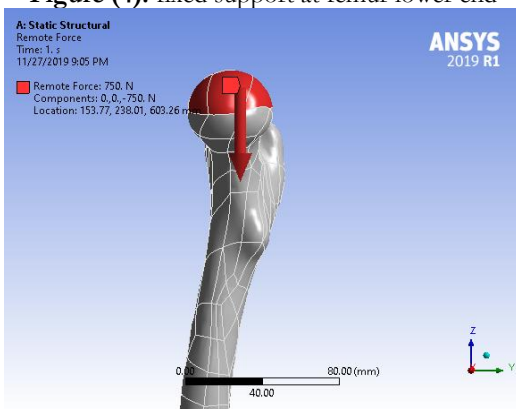


Figure (5): Load applied at femoral head

3. Results

The Von-Mises stress and Total deformation of the femur bone only is shown in Figures 6&7, and the Total deformation and Von-Mises stress of bone-

plate construct for 4th, 5th and 6th model is shown in Figures 8, 9 & 10.

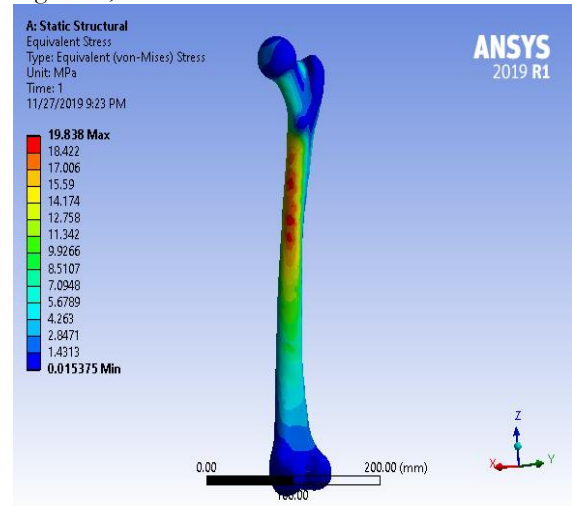


Figure (6): Femur bone equivalent stress.

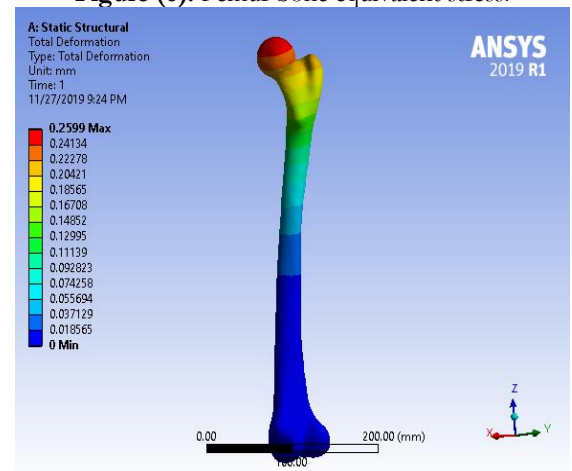


Figure (7): Femur bone total deformation

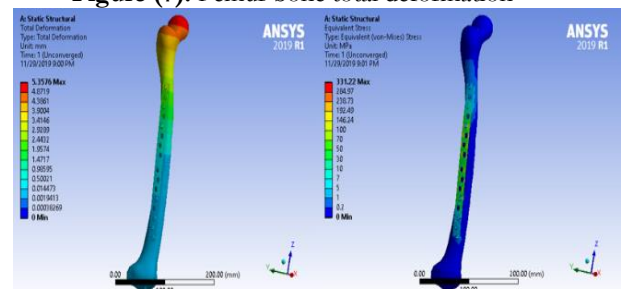


Figure (8): Bone-plate construct 4th model (with 10 screws) total Deformation and von-Mises stress.

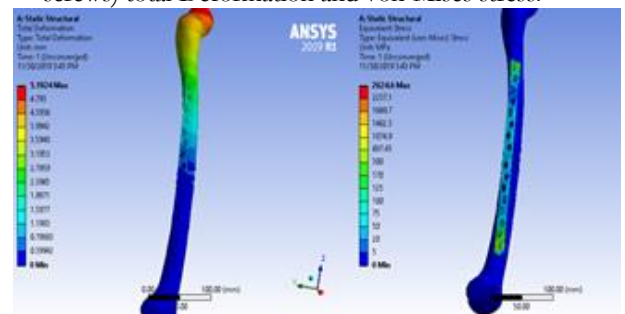


Figure (9): Bone-plate construct 5th model with 10 screws Total Deformation and von-Mises stress.

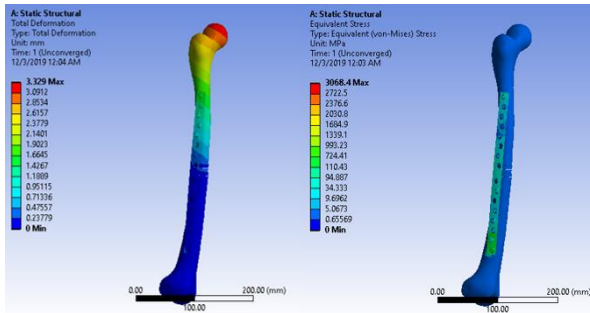


Figure (10): Bone-plate construct 6th model (with 12 screws) total Deformation, von-Mises stress.

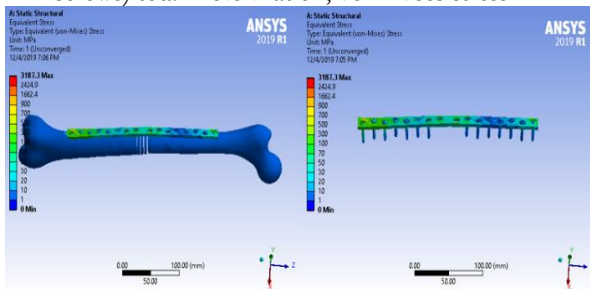


Figure (11): Plate and screws stress analysis 7th model (with 14 screws).

The seven models in table 1. Analyzed in ANSYS 2019 R1 Workbench for total Deformation and von-Mises stress and stated in table 3, from these tables curves are represented for the relations of total deformation, equivalent stress versus the number of screws models or the number of screws in the model. See Figures 12&13.

Table (3): The equivalent stress and total deformation with respect to models.

Model	Maximum equivalent stress MPa	Maximum deflection mm
2	255.55	7.6077
3	372.72	5.4301
4	331.22	5.3576
5	2624.6	5.1924
6	3068	3.329
7	26964	2.9453

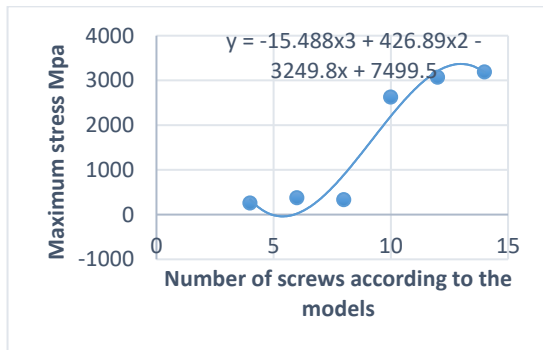


Figure (12): Bone-plate construct maximum von-Mises stress versus the number of screws according to the models.

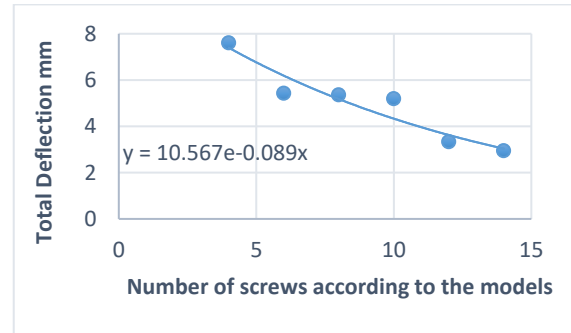


Figure (13): Bone-plate construct Total Deformation versus the number of screws according to the models.

The only 1st model is suffered a failure by loading conditions as shown in Figures 14&15, so it is not considered in calculations of the curves.

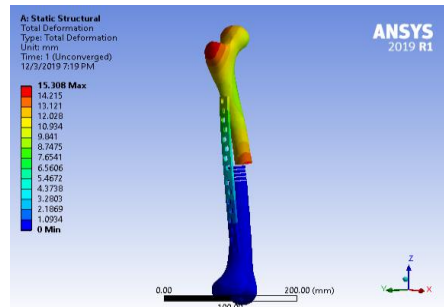


Figure (14): 1st model (with two screws) total deformation

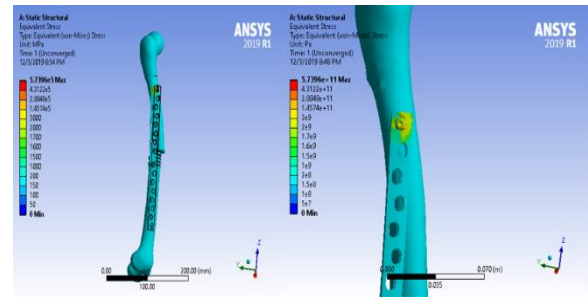


Figure (15): 1st model (with two screws) a) equivalent stress with undeformed plate planning, b) screw in the load destination.

The fracture gap displacement ΔL for the models, is stated in table 4. And from these results of the displacement Interfragmentary strain is calculated by using equation.eq.1.

Table (4): The fracture gap displacement and interfragmentary strain with respect to models.

Model	Body load 750 N ΔL (mm)	Interfragmentary strain
2	0.9755	0.036811
3	0.68032	0.025819
4	0.68662	0.026058
5	0.66042	0.025063
6	0.49647	0.018841
7	0.3814	0.014474

From table 1 which state the seven models of screws orientation, and table 4 which state the



interfragmentary strain, a curve of interfragmentary strain versus the models of screws is represented in Fig.16.

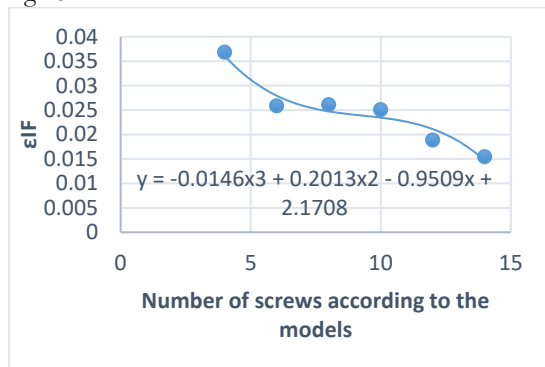


Figure (16): Interfragmentary strain versus the number of screws or models of screws orientation at load 750N

4. Discussion

From the curves of equivalent stress and total deformation versus the number of screws, Figure 10. Shows that the maximum stress in the bone-plate construct is increased by increasing the number of screws except in first model (one screw in each side of plate) is represented high stress due to failure due to loosening of plate and screws so the implant did not bear the load condition. And figure 10 also shows that increasing the number of screws leads to decrease the total deformation, and also the interfragmentary strain at the fracture gap decreased by increasing the number of screws. Perren's interfragmentary strain theory [10] stated that the magnitude of ϵ_{IF} determines the tissue differentiation at the fracture gap, because each tissue has different strain tolerance. According to Perren's strain theory ϵ_{IF} less than 0.2% form direct bone formation, less than 10% allow cartilage differentiation and subsequent endochondral ossification (indirect healing), and more than 10 % leads to nonunion, so from table 6. We can decided which the screw arrangement (model) is the best.

5. Conclusion

- ✚ The maximum stress increased with increasing the number of screws as polynomial equation and the total deformation decreased by increasing the number of screws as exponential equation curves.
- ✚ Interfragmentary strain of the gap could be decreased by increasing the number of screws as polynomial equation.
- ✚ First model undergo failure and not acceptable for fixation.
- ✚ Seventh model not much accepted in spite of the interfragmentary strain is less than 0.02 but the resulted stress is very high weakens the bone.
- ✚ 2nd-6th Models are acceptable according to Perren's strain theory.

6. References

[1] M. Gudarzi, H. Zamanian, B. Marzban, and S. Amoozegar, "Investigation of the Stress Distribution

in a Bone Due to Screws Pretensions of the Bone Plate," *Elastic*, vol. 20, pp. 0.3, 2012.

[2] M. Schimmelpfenning. "Patient specific optimization of fracture treatment considering the inhomogeneous material properties of bone tissue and the expected load situation," https://www.ozeninc.com/wpcontent/uploads/2014/12/03_WOST2013_Optimization_Schimmelpfenning_Paper.pdf.

[3] G. R. Kristina Haase, "Finite Element Analysis of Orthopaedic Plates and Screws to Reduce the Effects of Stress Shielding," Master of Applied Science in Mechanical Engineering University of Ottawa, Ontario 2009.

[4] B. Wongchai, "The Effect of the Configuration of the Screw Fixation on the Interfragmentary Strain," *American Journal of Applied Sciences*, vol. 9, pp. 842-845, 1/6/2012.

[5] J. K. Oh, D. Sahu, Y. H. Ahn, S. J. Lee, S. Tsutsumi, J. H. Hwang, et al, "Effect of fracture gap on stability of compression plate fixation: a finite element study," *J Orthop Res*, vol. 28, pp. 462-7, Apr 2010

[6] P. S. R. S. Maharaj, R. Maheswaran, and A. Vasanthanathan, "Numerical Analysis of Fractured Femur Bone with Prosthetic Bone Plates," *Procedia Engineering*, vol. 64, pp. 1242-1251, 2013.

[7] U. Mughal, H. Khawaja, and M. Moatamedi, "Finite element analysis of human femur bone," *The International Journal of Multiphysics*, vol. 9, no. 2, 2016.

[8] D. Amalraju, and A. S. Dawood, "Mechanical strength evaluation analysis of stainless steel and titanium locking plate for femur bone fracture," *Engineering Science and Technology: An International Journal*, vol. 2, no. 3, pp. 381-388, 2012.

[9] S. Das, and S. K. Sarangi, "Finite element analysis of femur fracture fixation plates," *International Journal of Basic and Applied Biology*, vol. 1, no. 1, pp. 1-5, 2014.

[10] P. K. Satapathy, B. Sahoo, L. N. Panda, and S. Das, "Finite element analysis of functionally graded bone plate at femur bone fracture site," *IOP Conference Series: Materials Science and Engineering*, vol. 330, p. 012027, 2018.